IN-39
157895
p. 58

NASA Technical Memorandum 106001

High Temperature Composite Analyzer (HITCAN) Theoretical Manual

Version 1.0

J.J. Lackney
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park, Ohio

and

P.L.N. Murthy and P.K. Gotsis
Lewis Research Center
Cleveland, Ohio

April 1993

(NASA-TM-106001) HIGH TEMPERATURE COMPOSITE ANALYZER (HITCAN) THEORETICAL MANUAL, VERSION 1.0 (NASA) 58 p

N93-24913

Unclas

G3/39 0157895

NASA

TABLE OF CONTENTS

Page

ABSTRACT ABST1

1. INTRODUCTION INTRO1

2. METCAN-MHOST SYNTHESIS SYNTH1

   2.1 Nonlinear Analysis SYNTH3

   2.2 Composite Analysis SYNTH7

3. CAPABILITIES AND LIMITATIONS CAP/LIM1

REFERENCES REF1

APPENDICES

1. METCAN THEORY MET/THEORY1

   1.1 Composite Micromechanics Theory COMP.MIC/THEORY1

   1.2 Laminate Theory LAM/THEORY1

   1.3 Material Model and Fiber Degradation MAT/FIB/DEGRAD1

2. MHOST THEORY MHOST/THEORY1

   2.1 Displacement Formulation DISP/FORM1

   2.2 Elements ELEM1

   2.3 Buckling and Modal Analysis BUCK/MOD/ANAL1

ABSTRACT

This manual outlines some of the theoretical aspects embedded in the computer code, HITCAN (HIgh Temperature Composite ANalyzer). HITCAN is a general purpose computer program for predicting nonlinear global structural and local stress-strain response of arbitrarily oriented, multilayered high temperature metal matrix composite structures. This code combines composite mechanics and laminate theory with an internal data base of the constituents (matrix, fiber and interface) material properties. The thermal and mechanical properties of the constituents are considered to be nonlinearly dependent on several parameters including temperature, stress and stress rate. The computational procedure for the analysis of the composite structure uses the finite element method. HITCAN consists of three modules: COBSTRAN, METCAN and MHOST. COBSTRAN generates the geometry (pre-processor) and defines the lay up of the different plies. METCAN computes the material behavior of the composite and of the constituents. Finally, MHOST is a finite element program based on the mixed iterative solution technique. MHOST has a library for 2D and 3D isoparametric elements. HITCAN is written in FORTRAN 77 computer language and has been configured and executed on the NASA Lewis Research Center CRAY XMP and YMP computers.

CHAPTER 1

INTRODUCTION

HITCAN (High Temperature Composite ANalyzer) is a general purpose computer program for predicting the global structural response of high temperature metal matrix composite structures. This document describes the basic theory behind this code.

The HITCAN computer program is a combination of three computer codes developed in-house at the NASA Lewis Research Center. They are:

- a mesh generator; COBSTRAN (Reference 1);
- a metal matrix composite analyzer; METCAN (Reference 2);
- a finite element structural analysis code; MHOST (Reference 3).

In Chapter 2, the theoretical aspects of the METCAN-MHOST synthesis will be discussed. The capabilities and limitations of HITCAN are discussed in Chapter 3. In Appendices 1 and 2, the theory behind the individual library modules METCAN and MHOST will be reviewed. Since the functionality of COBSTRAN as used in HITCAN, is restricted to only the mesh generation, its theoretical aspects will not be discussed here. The reader is referred to Reference 4 for further information.

CHAPTER 2

METCAN-MHOST SYNTHESIS

As mentioned previously, HITCAN is a computer program for predicting the global structural response of high temperature metal matrix composite structures. At high temperature, metal matrix composite structures experience nonlinear material behavior. As a result of this nonlinear behavior, the finite element equations are nonlinear. Consequently, for HITCAN to be of any use it must be able to solve these equations. Thus, a nonlinear solution scheme was incorporated into HITCAN.

The nonlinear solution scheme consists of two parts. The first part is an analysis of the composite material at the laminate, ply, and constituent levels. To perform this local analysis, METCAN was chosen. The second part of the scheme is a global analysis of the structure using MHOST. Thus, the core of this nonlinear solution scheme is the METCAN-MHOST synthesis. Both METCAN and MHOST are stand-alone computer programs. Hence, it was desirable that neither one of these programs would be modified extensively in order to work in HITCAN. In order to minimize the number of modifications, it was decided that the finite element formulation for linear structural analysis would serve as the basis for nonlinear structural analysis. For the nonlinear analysis, the linear formulation is applied in an incremental/iterative manner. Before proceeding further, a brief review of the finite element formulation for linear analysis is in order.

The global formulation of a linear static structural analysis problem using the finite element method is given in the following expression:

$$[K]\{u\} = \{F\}$$

In this expression, [K] is the global structural stiffness matrix, {u} is the vector of nodal displacements, and {F} is the vector containing the applied nodal loads.

The next expression is the eigenvalue problem corresponding to a free vibration analysis.

$$([K] - \lambda[M])\{u\} = \{0\} \Rightarrow \{\omega_n\}$$

In this expression, [K] is the global structural stiffness matrix, [M] is the global structural mass matrix, λ is the eigenvalue associated with the solution of the problem, [I] is the identity matrix, and $\{\omega_n\}$ is vector containing the natural frequencies where n is the number of modes.

The last expression is the eigenvalue problem corresponding to a static buckling analysis.

$$([K] + \lambda[K_\sigma])\{u\} = \{0\} \Rightarrow \{S_{cr}\},$$

where $[K_\sigma]$ is the stress stiffness matrix, [K] is the linear stiffness matrix, λ some scaler multiplier of $[K_\sigma]$, {0} is the null column vector, and $\{S_{cr}\}$ is the critical buckling load.

Although the formulation summarized above is for a general linear structural analysis problem, it can be applied in the nonlinear analysis of a structure. How this can be accomplished is described in the following section.

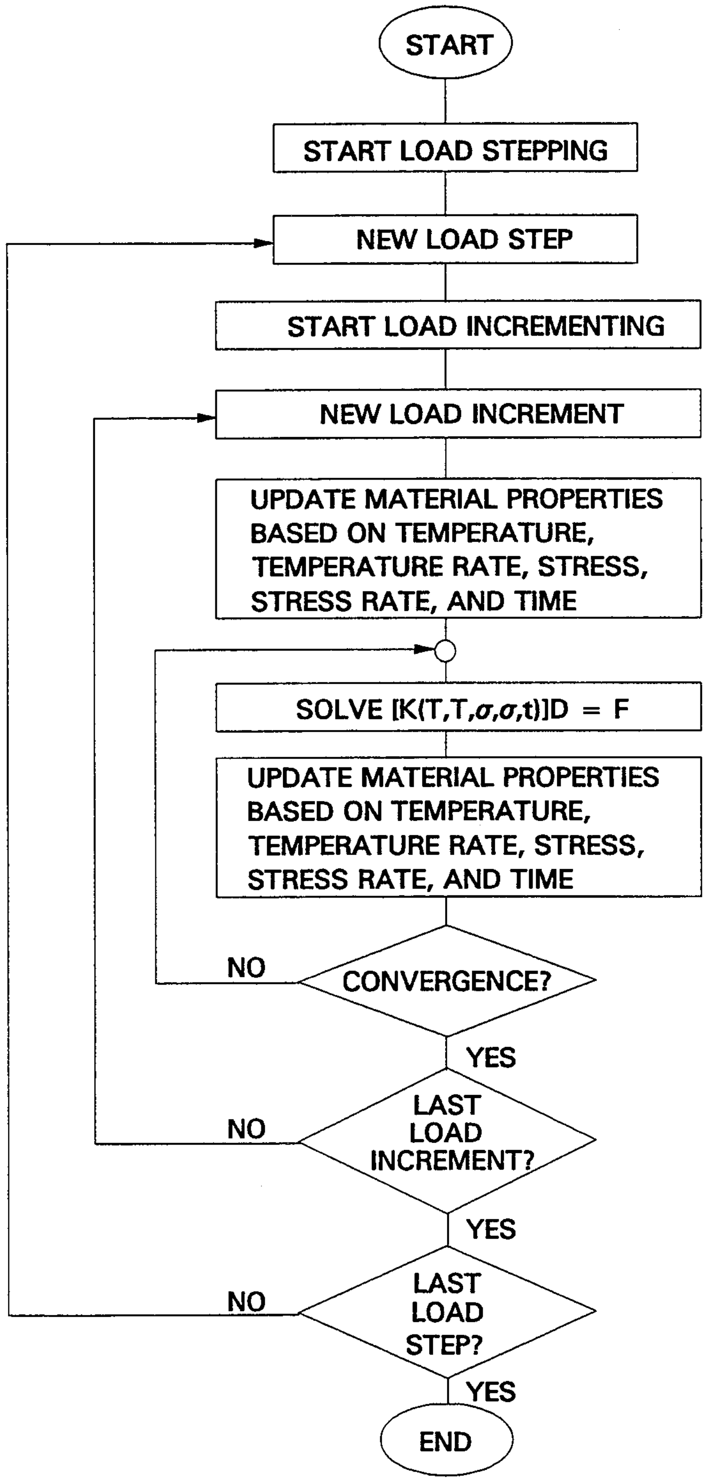
2.1 NONLINEAR ANALYSIS

The finite element formulation for a linear structural analysis can serve as the basis for a nonlinear structural analysis. For a nonlinear analysis, the linear formulation is applied in a incremental/iterative manner. This approach allows for the fact that the material and thermal properties are not constant over the range of the problem, but are actually dependent functions of the primitive variables, such as stress, rate of stress, temperature, rate of temperature, metallurgical reaction between fiber and matrix, and time.

In applying this incremental/iterative approach, the solution of a nonlinear structural analysis problem is treated as an iterative process that involves solving the linear problem formulation for distinct load increments of the total range of the solution for the problem. Furthermore, as the iterative solution process progresses, the results from the analysis of each step or increment are used to update the material and thermal properties of the composite at each successive increment. The flow diagram for the incremental/iterative approach is shown in Figure 2.1.

In HITCAN, the iterative procedure chosen was the direct iteration method. Assuming a single degree-of-freedom system and that the stiffness is a function of the temperature, temperature rate, stress, and stress rate, the equation that needs to be solved is $[K(T,T,\sigma,\sigma,t)]D = F$. The direct iteration method works as follows.

- Initially, the material properties are calculated based on the nodal temperature, temperature rate, stresses, and stress rates obtained from the previous load increment and time.



INCREMENTAL/ITERATIVE APPROACH
FIGURE 2.1

SYNTH4

- The incremental displacements are calculated from the equation

$$\Delta D^1 = [K(T,T,\sigma,\sigma,t)^0]^{-1}\Delta F,$$

where $[K(T,T,\sigma,\sigma,t)^0]$ is the initial slope and ΔF is the incremental load.

Repetition of this process can be written as

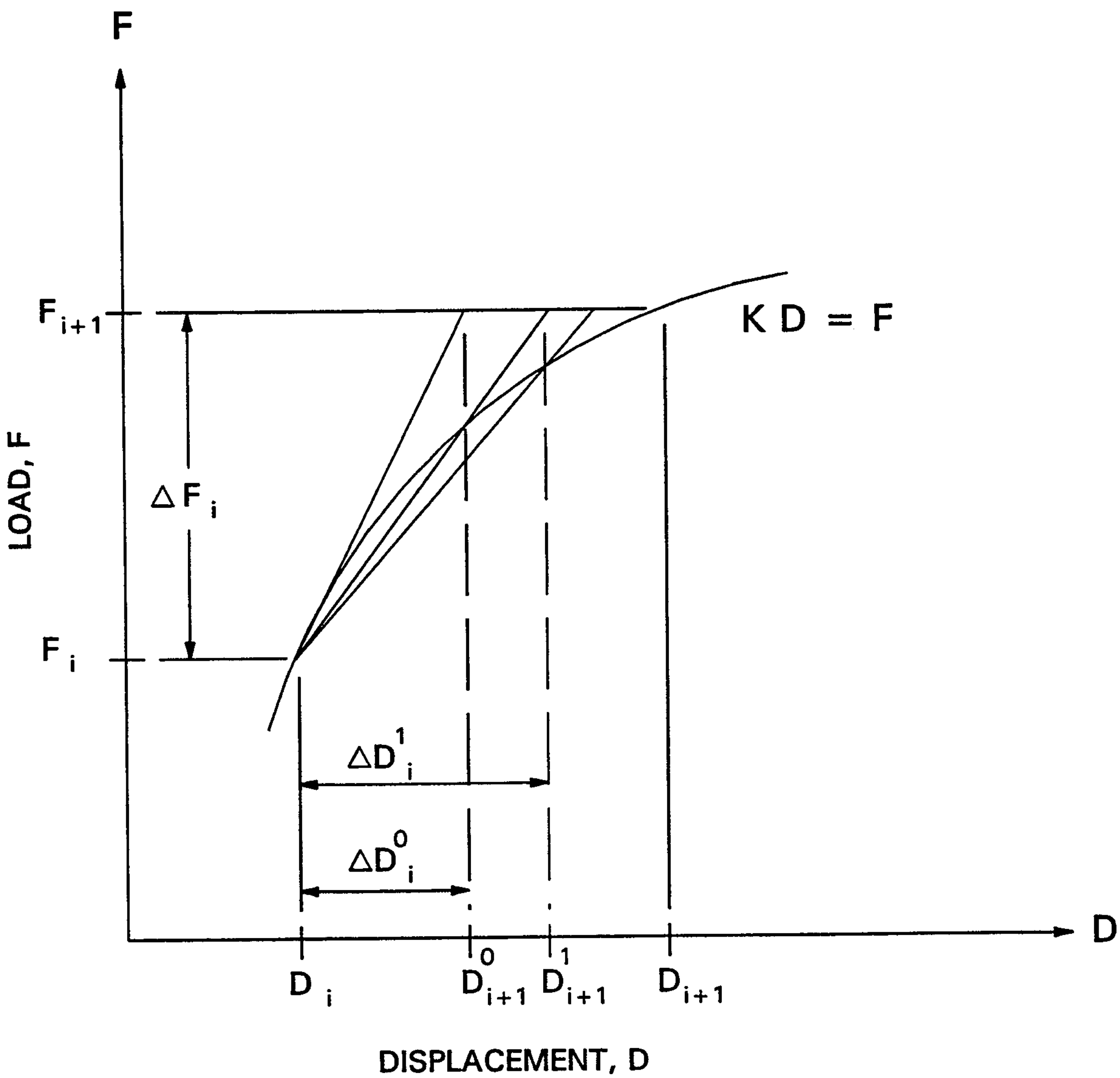
$$\Delta D^n = [K(T,T,\sigma,\sigma,t)^{n-1}]^{-1}\Delta F.$$

Here the slope $[K(T,T,\sigma,\sigma,t)^{n-1}]$ is calculated based on the material properties of the previous iteration. The material properties are functions of the nodal temperature, temperature rate, stresses, and stress rate.

- This process is terminated when the error becomes sufficiently small, i. e. ,

$$||\Delta D^n - \Delta D^{n-1}||/||\Delta D^{n-1} - \Delta D^{n-2}|| \leq \text{tolerance}.$$

This process is shown graphically in Figure 2.2.



DIRECT ITERATION METHOD

FIGURE 2.2

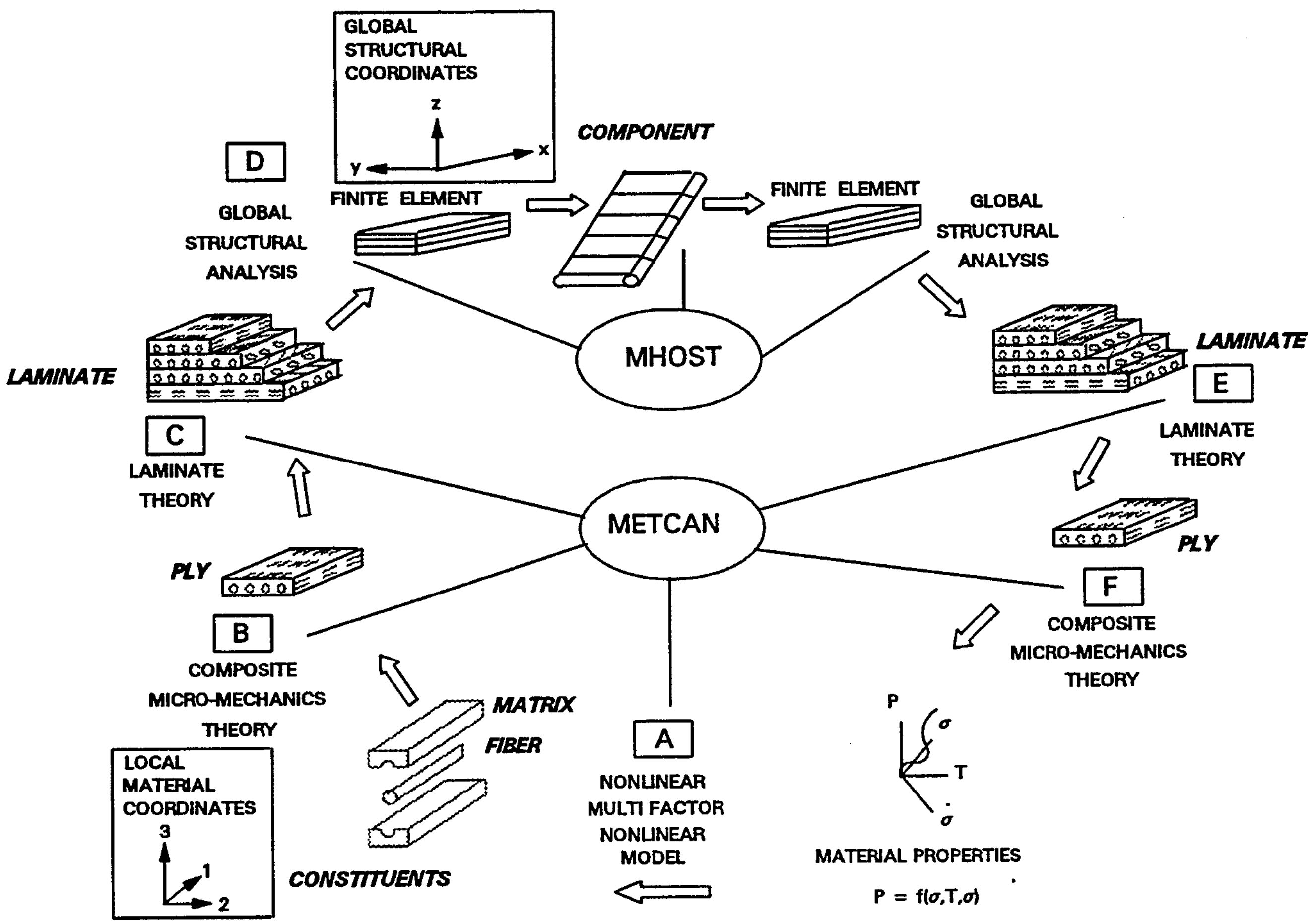
2.2 COMPOSITE ANALYSIS

The distinguishing features involved in the analysis of a composite structure include the additional theories and assumptions employed to represent the inherently heterogeneous composite structure as an equivalent "pseudo-homogeneous" structure. In the approach taken in HITCAN, these additional theories provide for the generation of the various global structural properties and for the degeneration of the material properties during the incremental solution process. These aspects, unique to the analysis of a composite structure, involve such subjects as composite micromechanics theory and laminated plate theory.

The iterative approach used in HITCAN, as illustrated in Figures 2.1 and 2.2, is shown schematically in Figure 2.3. As can be seen in the figure, the direct iteration method has been modified to account for the heterogeneity of composite materials. The key elements comprising this analysis include:

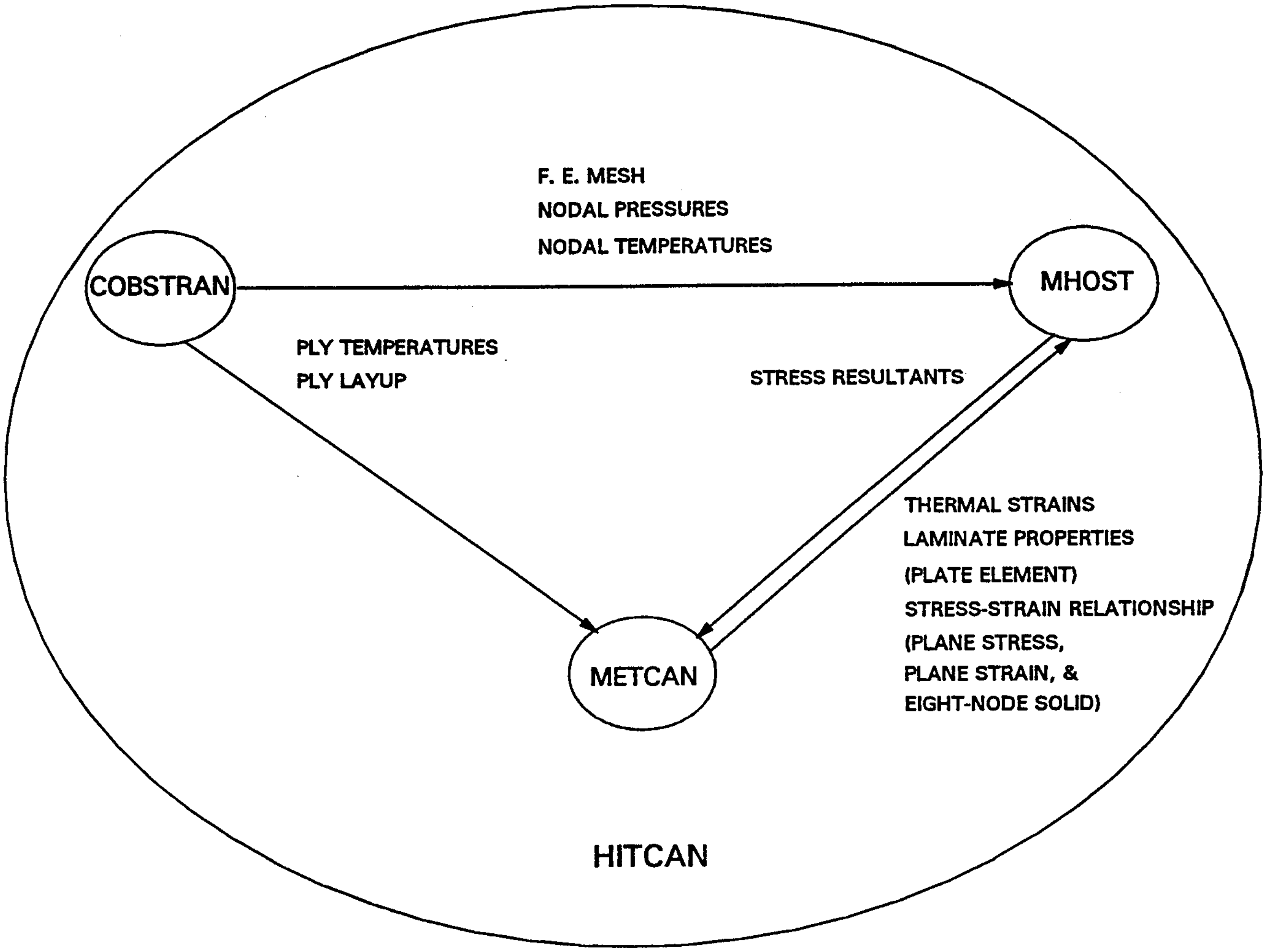
- multifactor interaction model for nonlinear material;
- composite micromechanics theory;
- laminate theory;
- finite element structural analysis.

The circular arrangement of Figure 2.3 is intended to illustrate the iterative solution strategy imbedded in the incremental/iterative approach as it is used in HITCAN. Figure 2.4 shows the quantities passed between METCAN and MHOST, namely, stress resultants, laminate properties, thermal strains, and composite properties.



HITCAN: AN INTERATIVE APPROACH FOR HIGH TEMPERATURE COMPOSITE STRUCTURAL ANALYSIS

FIGURE 2.3



METCAN-MHOST SYNTHESIS
FIGURE 2.4

As can be seen in the Figure 2.3, the two library modules METCAN and MHOST comprise the iterative process. In this scheme, iteration is required at two levels. The first or local level is in METCAN. This is required since, in the nonlinear material model, the constituent properties are functions of the constituent temperature, temperature rates, stresses, stress rates, and time. The second or global iteration is required to establish equilibrium of the global structural response.

From Figure 2.3, it can be seen that METCAN performs steps E, F, A, B, and C; i. e.,

- METCAN takes the stress resultants obtained from the structural response and determines the ply stresses, using laminate theory;

- Using composite micromechanics theory METCAN determines the constituent stresses and stress rates;

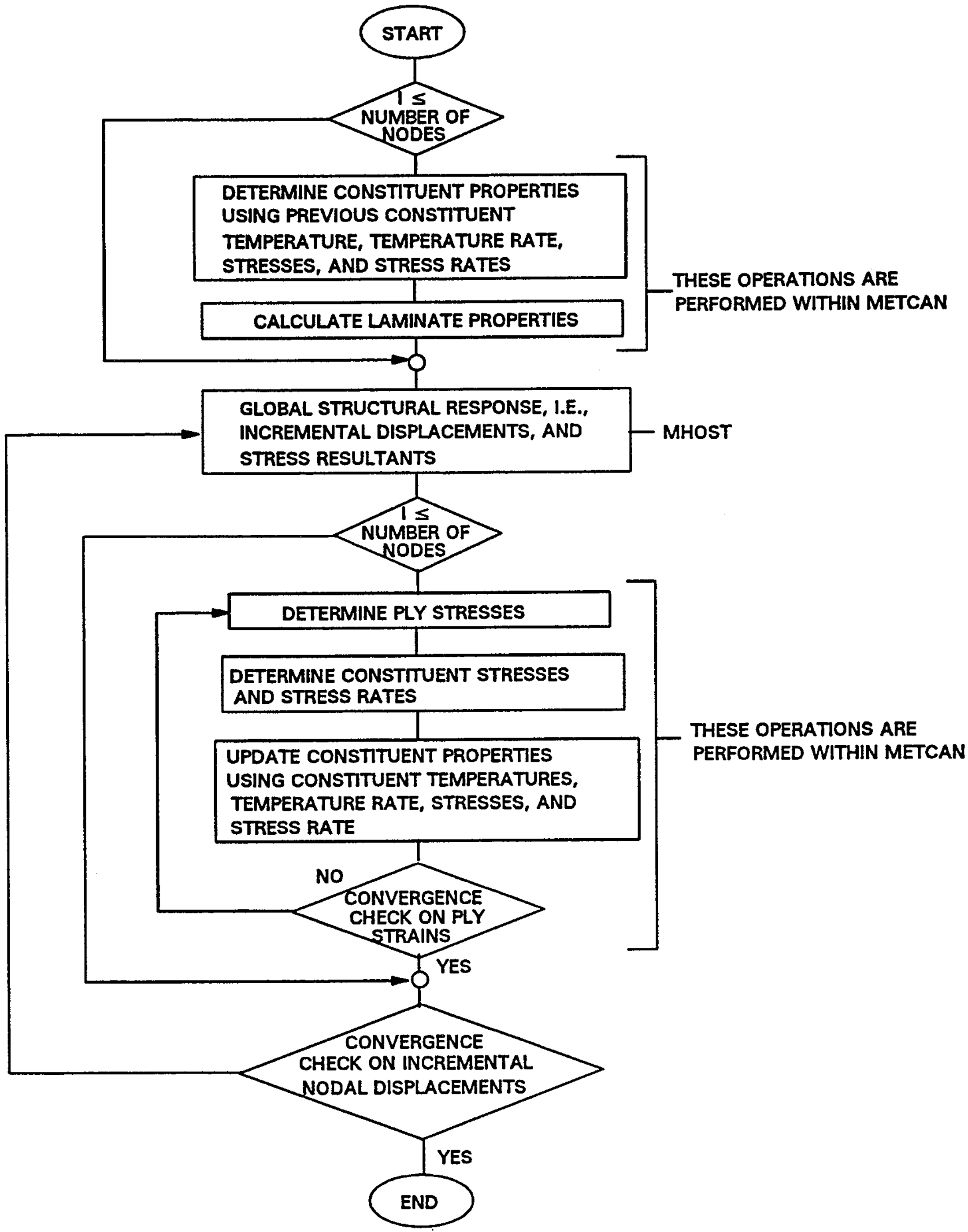
- METCAN then calculates new constituent material properties using the multi-factor model;

- From micromechanics theory, METCAN is then able to calculate new ply properties;

- With the new ply properties, METCAN then determines the new laminate properties;

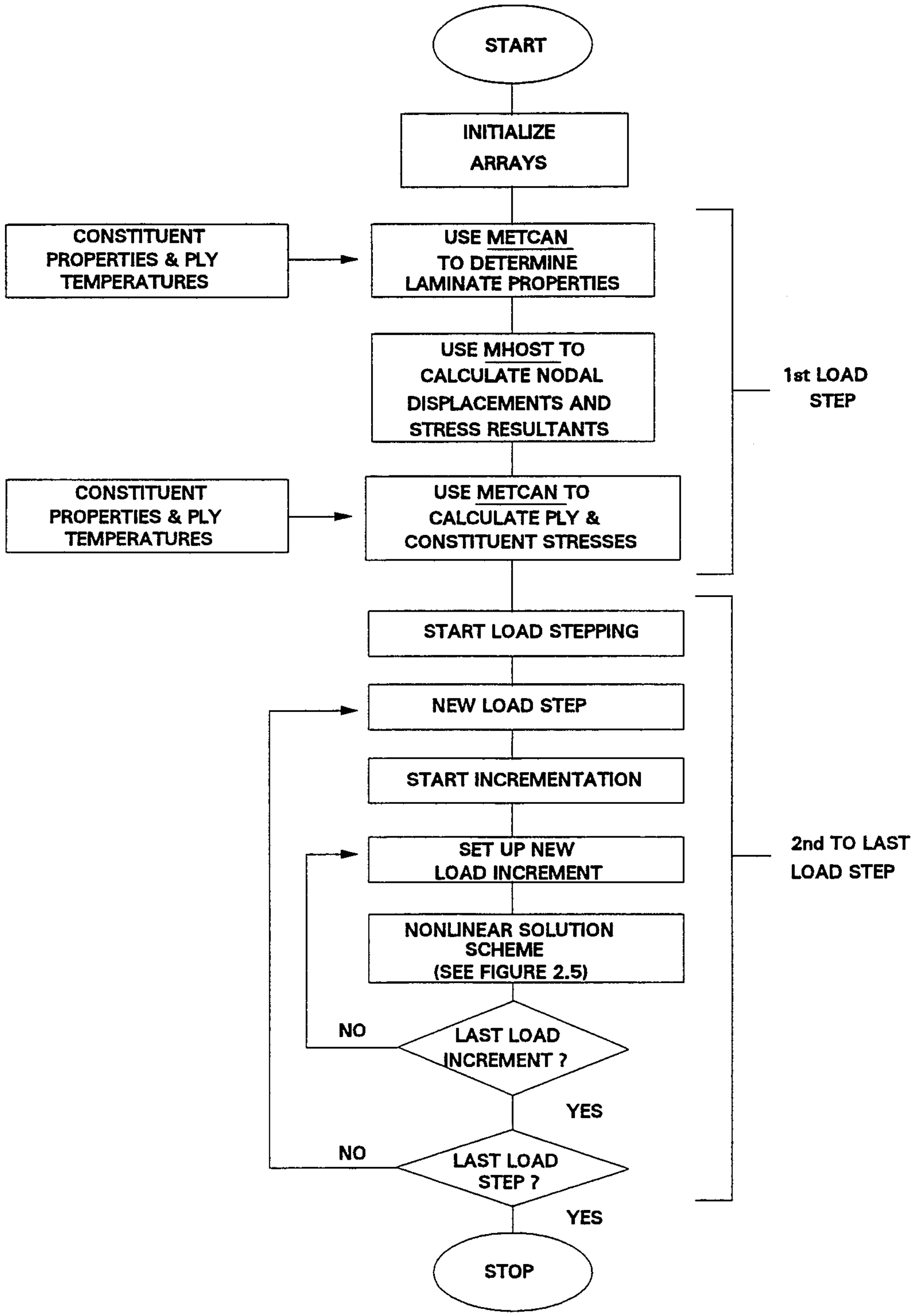
- Using laminate theory, METCAN calculates the strains at each node. Using these ply strains, convergence is checked within METCAN.

The iterative process occurring in METCAN comprises the steps E, F, A, B, and C. Since these steps are contained entirely within METCAN, it is called only once for each node. As shown in Figure 2.3, MHOST simply determines the global response. If a convergence criteria is included in the loop, Figure 2.3 can be redrawn as shown in Figure 2.5. Figure 2.6 shows the flow diagram of the incremental/iterative process used in HITCAN (subroutine NLINER).



DIRECT ITERATION METHOD AS USED IN HITCAN

FIGURE 2.5



GENERAL FLOW DIAGRAM OF SUBROUTINE NLINER
FIGURE 2.6

CHAPTER 3

CAPABILITIES AND LIMITATIONS

HITCAN is capable of predicting global structural and local stress-strain response of multilayered HTMMC structures exhibiting nonlinear material behavior. The constitutive model employed in METCAN is specifically designed for HTMMC, therefore it is recommended that HITCAN be used only for metal matrix composites.

At the present time, the following analyses are available in HITCAN:

- Incremental static analysis with nonlinear anisotropic material behavior;
- Dynamic analysis using direct time integration;
- Modal analysis (free vibration);
- Buckling analysis (first critical buckling load).

The element library includes 3 four-noded elements, i.e. plate, plane stress, and plane strain, and 1 eight noded element, i.e. a 3D solid. The current mesh generation capability of the code allows modeling of solid structures using any of the 4 types of elements. The user may also input a finite element model directly. The code is capable of handling a variety of boundary conditions, loadings (centrifugal, concentrated, distributed, pressure, temperature, static, transient, cyclic, and impact), and various types of structures (such as beam, plate, ring, curved panel, and built-up structures). A list of HITCAN's capabilities can be found in Table 3.1.

The limitations of the code are :

- Formulation assume small displacement and small strain theory;
- Elements of different types cannot be combined;
- Hollow structures can be modeled using the plate element only;
- The finite element model generated by HITCAN can have a nonuniform mesh only along the x-axis;
- If the curvature is large, the mesh will not be uniform.

HITCAN presents analysis results at the global, laminate, and ply levels. Results include displacements, reactions, stresses, and modes shapes. The code also has the capability to generate post-processing files for PATRAN.

Type of Element → / Type of Analysis ↓	Plate	Plane Strain	Plane Stress	8-Node Solid
Static	tested	-	-	tested
Buckling (a)	tested	-	-	-
Load Stepping	tested	-	-	tested
Modal (Natural Vibration Modes) (b)	tested	-	-	-
Time-domain	-	-	-	-
Loading				
Mechanical	tested			tested
Thermal	tested			tested
Cyclic	-	-	-	-
Impact	-	-	-	-
Constitutive Models (c)				
$P = \text{Constant}$	tested	-	-	tested
$P = f(T)$ (temperature dependence)	tested	-	-	tested
$P = f(\sigma)$ (stress dependence)	tested	-	-	tested
$P = f(\sigma)$ (stress rate dependence)	tested	-	-	tested
$P = f(t)$ (creep)	-	-	-	-
$P = f(T\dot{\sigma}, \sigma, \sigma)$ (combination)	tested	-	-	tested
$P = f(T\sigma, \dot{\sigma}, \sigma, t)$ (creep combination)	-	-	-	-
Fiber Degradation	tested	-	-	tested
Fabrication-induced Stresses	tested	-	-	tested
Ply Orientations				
Arbitrary	tested	-	-	tested

(a) Tested 1 buckling mode
(b) Tested 4 vibration modes

(c) Constitutive models: Notation
P: Material properties
T: Temperature
σ: Stress
$\dot{\sigma}$: Stress rate
t: Time

TABLE 3.1: HITCAN Capabilities for Composite Materials

REFERENCES

1. Aiello, R. A.,"Composite Blade Structural Analyzer User's Manual", NASA, 1989.

2. Lee, H. J., Gotsis, P. K., Murthy, P. L. N. and Hopkins D. A.,"Metal Matrix Composite Analyzer User's Guide", NASA, 1988.

3. Nakazawa, S.,"The MHOST Finite Element Program: 3-D Inelastic Analysis Methods for Hot Section Components", Volume II, NASA, 1989.

4. Aiello, R. A.,"Composite Blade Structural Analyzer Theoretical/Programmer's Manual", NASA, 1989.

5. Hopkins, D. A.,"Nonlinear Analysis for High Temperature Multilayered Fiber Composite Structures", NASA TM-83754, 1984.

6. Hopkins, D. A. and Chamis, C. C.,"A Unique Set of Micromechanics Equations for High Temperature Metal Matrix Composites", NASA TM-87154, 1985.

7. Chamis, C. C. and Hopkins, D. A.,"Thermoviscoplastic Nonlinear Constitutive Relationships for Structural Analysis of High Temperature Metal Matrix Composites", NASA TM-87291, 1985.

8. Chamis, C. C., Murthy, P. L. N., and Hopkins, D. A.,"Computational Simulation of High Temperature Metal Matrix Composite Cyclic Behavior", NASA TM-102115, 1988.

9. Murthy, P. L. N., Hopkins, D. A., and Chamis, C. C., "Metal Matrix Composite Micromechanics: In-Situ Behavior Influence on Composite Properties", NASA TM-102302, 1989.

10. Chamis, C. C., Caruso, C. J., Lee, H.-J., and Murthy, P. L. N.,"METCAN Verification Status", NSAS TM-103119, 1990.

11. Lee, H.-J., Murthy, P. L. N., and Chamis, C. C.,"METCAN Updates for High Temperature Composite Behavior: Simulation/Verification", NASA TM-103682, 1991.

12. Ashton, J. E. and Whitney, J. M., Theory of Laminated Plates, Tecnomic Publishing Co., Inc., Stanford, Connecticut, 1970.

13. Cook, R. D., Concepts and Applications of Finite Element Analysis, John Wiley and Sons, Inc., 1981.

14. Bathe, K.-J., Finite Element Procedures In Engineering Analysis, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1982.

15. Hughes, T. R., The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1987.

APPENDIX 1

METCAN THEORY

In this chapter, the theoretical aspects of METCAN will be reviewed. Only a brief review can be given here. For more detailed information the reader is directed toward References 5 - 10.

As stated in the previous chapter, METCAN performs the following iterative functions in HITCAN:

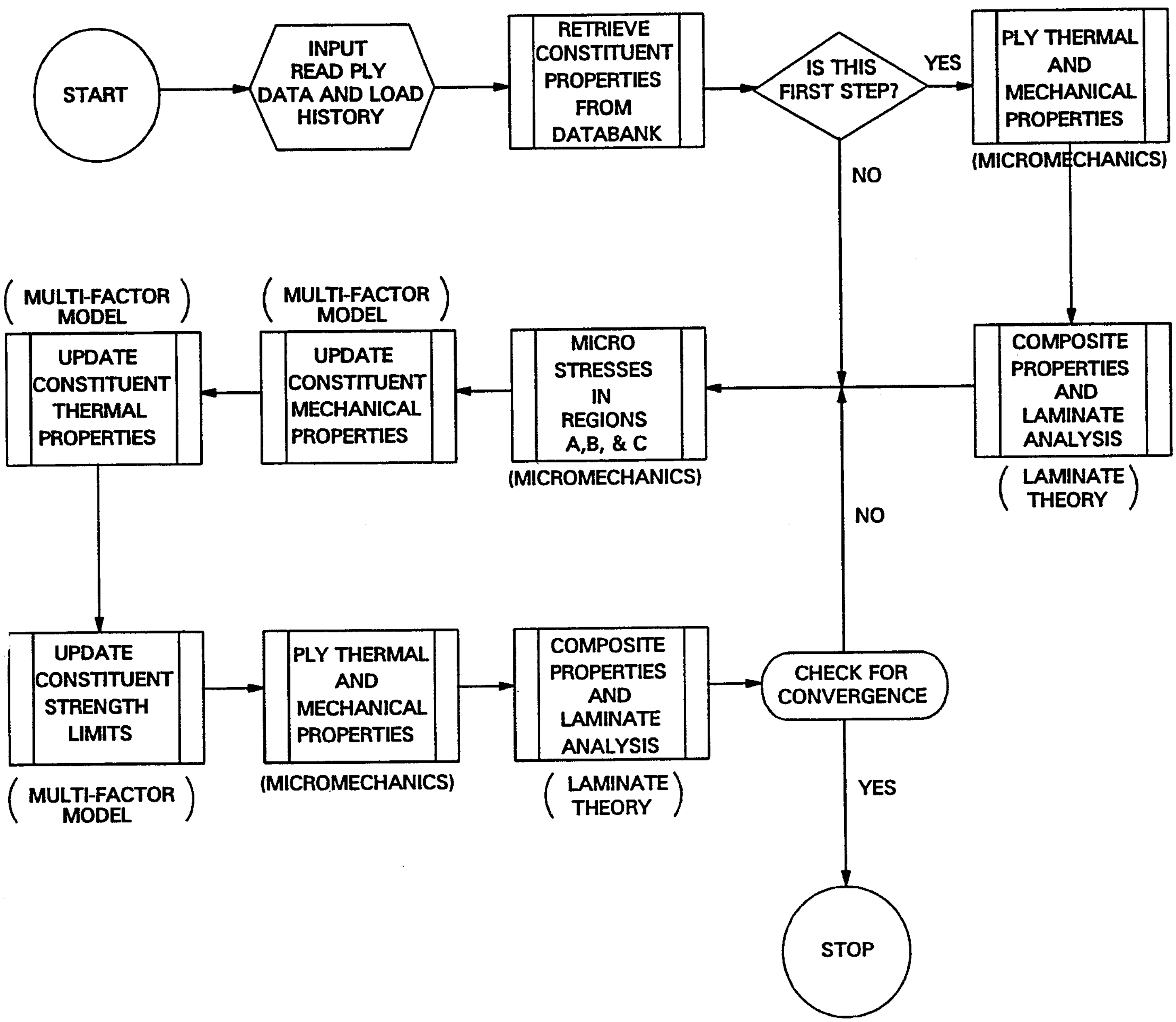
- METCAN takes the stress resultants obtained from the structural response and determines the ply stresses, using laminate theory;
- using composite micromechanics theory METCAN determines the constituent stresses;
- METCAN then calculates new constituent material properties using the multi-factor model;
- from micromechanics theory, METCAN is then able to calculate new ply properties;
- with the new ply properties, METCAN then determines the new laminate properties;
- using laminate theory METCAN, calculates the ply strains at each node. These strains are used in checking convergence.

It was also stated previously that because of the form of the nonlinear constitutive model, iterating is required within METCAN. With all of the above in mind, the flow diagram for METCAN can be drawn as shown in Figure A1.1. For the check on convergence, METCAN uses the ply strains; i. e.,

$$(\epsilon_i^{new} - \epsilon_i^{old})/\epsilon_i^{old} \leq \text{tolerance},$$

where i = 1,...,6 at each ply.

The rest of this chapter is devoted to the key elements of METCAN, namely, composite micromechanics theory, laminate theory, nonlinear constitutive model, and fiber degradation. Composite micromechanics theory can be found in Appendix 1.1. In Appendix 1.2, a brief review of laminate theory is given and in Appendix 1.3 the constitutive model and fiber degradation are discussed.



METCAN FLOW DIAGRAM

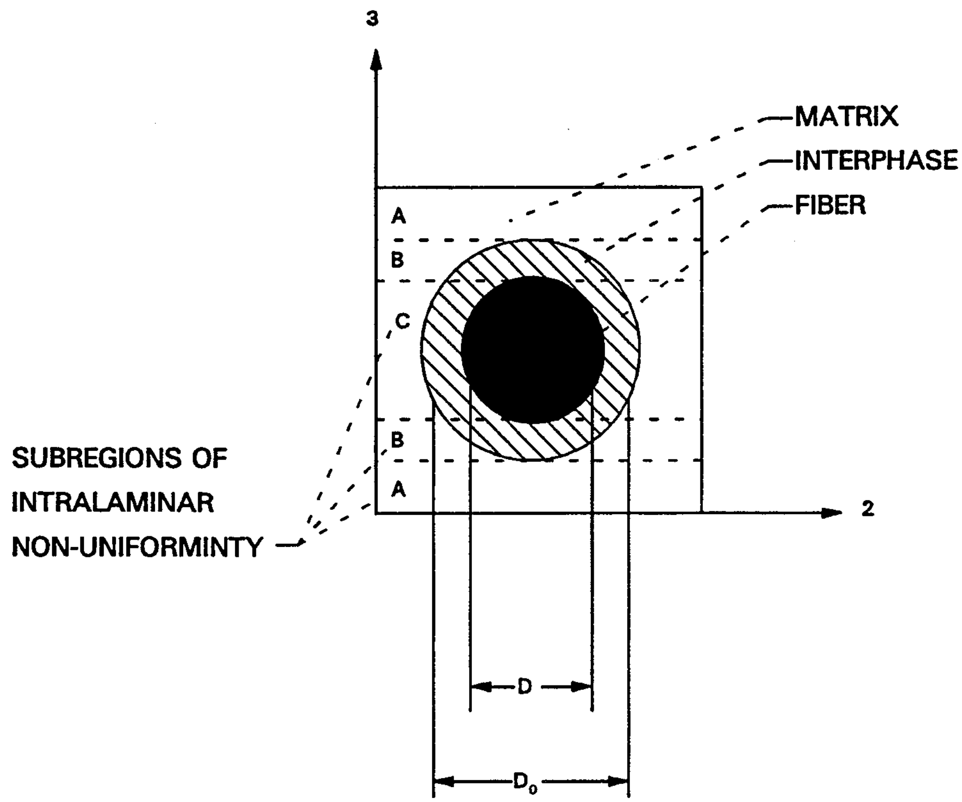
FIGURE A1.1

APPENDIX 1.1 - COMPOSITE MICROMECHANICS THEORY

Composite micromechanics theory relies on the principles of solid mechanics, thermodynamics, etc. at different levels of mathematical sophistication, together with certain assumptions and approximations. In the approach taken in METCAN, the principles of displacement compatibility, force equilibrium, and Fourier's law for heat conduction are used. In addition, the following assumptions are made:

- fibers are continuous and parallel;
- properties of all fibers are identical;
- complete bonding exists between constituents.

The periodic structure of an unidirectional metal matrix ply is approximated by a square unit cell model. The geometry of the model is shown in Figure A1.2. It should be noted that the interphase growth is assumed to result from the degradation of fiber material and thus propagates inward causing a continuous decrease of the current fiber diameter from the original fiber diameter. With the existence of the interphase, three subregions (A, B, C) are distinguished to characterize the intralaminar (through-the-thickness) nonuniformity of the constituent microstresses and material properties. Further information on the composite micromechanics can be found in Reference 5.



MICROMECHANICS MODEL; SQUARE ARRAY UNIT CELL

FIGURE A1.2

APPENDIX 1.2 - LAMINATE THEORY

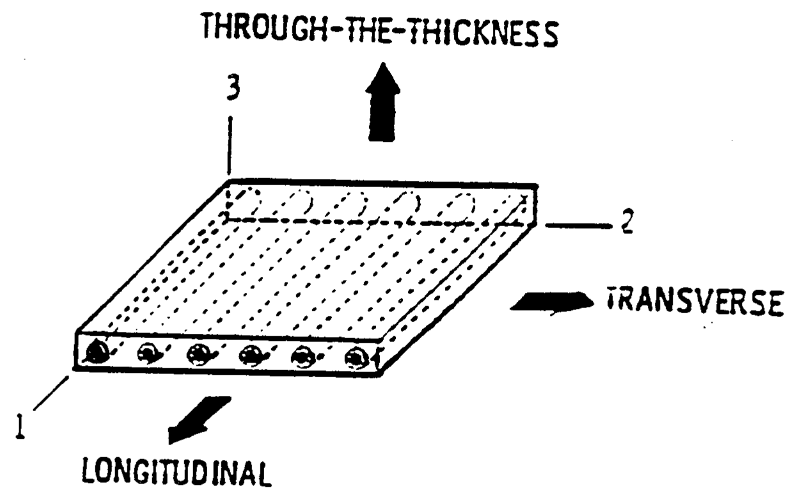
In this section, a brief review of laminate theory will be given. For a more complete discussion of laminate theory, the reader is referred to Reference 12.

Laminate theory provides a formal procedure to relate the behavior of a laminated composite structure to the behavior of the individual laminate of the composite. An unidirectional fiber-reinforced lamina can be approximated as a homogeneous orthotropic ply. From assumptions analogous to the Kirchoff hypothesis in classical plate theory, the laminae of a laminated composite structure are treated as thin plates and therefore are assumed to exist in a plane stress state. As such, the constitutive relationship for the k^{th} specially orthotropic ply (referred to the lamina material coordinate system as shown in Figure A1.3) is given by

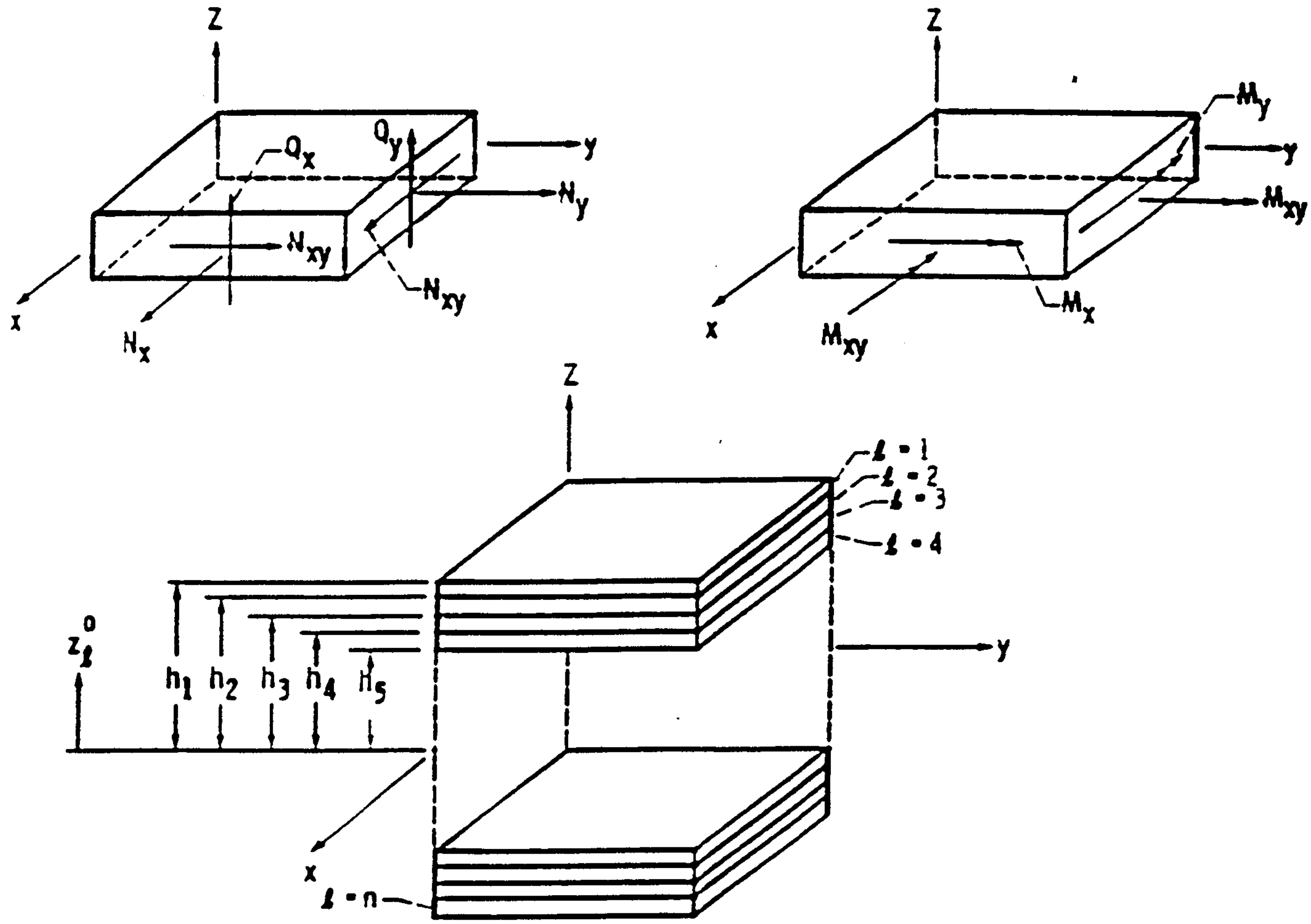
$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{12} \end{Bmatrix}_k = \begin{bmatrix} E_{11} & E_{12} & 0 \\ E_{12} & E_{22} & 0 \\ 0 & 0 & E_{66} \end{bmatrix}_k^{-1} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix}_k$$

In terms of stress, this can be rewritten as

$$\{\sigma_{\ell m}\}_k = [E_{\ell m}]_k \{\varepsilon_{\ell m}\}_k$$



UNIDIRECTIONAL COMPOSITE (PLY) MATERIAL COORDINATE SYSTEM
FIGURE A1.3



DEFINITION OF LAMINATE MIDPLANE STRESS AND MOMENT RESULTANTS
AND STRUCTURAL COORDINATE SYSTEM

FIGURE A1.4

where the subscript m indicates ply material coordinate system. For the generally orthotropic ply (referred to in the global structural coordinate system) the constitutive relationship is given by;

$$\{\sigma_{\ell s}\}_k = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix}_k = [R_{\ell\varepsilon}]_k^T \, [E_{\ell m}]_k \, [R_{\ell\varepsilon}]_k \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix}_k$$

$$= [E_{\ell s}]_k \, \{\varepsilon_{\ell s}\}_k$$

where the subscript s indicates global structural coordinate system. The matrix $[R_{\ell\varepsilon}]_k$ is the rotation or transformation matrix and is defined as;

$$[R_{\ell\varepsilon}]_k = \begin{bmatrix} \cos^2\theta & \sin^2\theta & \frac{1}{2}\sin 2\theta \\ \sin^2\theta & \cos^2\theta & -\frac{1}{2}\sin 2\theta \\ -\sin 2\theta & \sin 2\theta & \cos 2\theta \end{bmatrix}_k$$

and the angle θ represents the orientation of the lamina material coordinate axes (1, 2) for the k^{th} ply with respect to the global structural coordinate axes (x, y).

In terms of the midplane strain and laminate curvature, the constitutive relationship for the k[th] generally orthotropic lamina becomes;

$$\{\sigma_{\ell s}\}_k = [E_{\ell s}]_k \{\varepsilon_{\ell s}\}_k = [E_{\ell s}]_k (\{\varepsilon_c^0\} + z_\ell^0\{\kappa_c\})$$

where the subscript c denotes composite and z°_l is the perpendicular distance from the laminate midplane to the center of the k[th] ply.

The stress resultants (N_x, N_y, N_{xy}, M_x, M_y, M_{xy}) are shown in Figure A1.4. The resultants are given by the following equations;

$$\{N_c\} = \begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \int_{-h/2}^{h/2} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} dz$$

$$\{M_c\} = \begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \int_{-h/2}^{h/2} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} z\,dz$$

The single continuous integrals above can be separated into sums of integrals over each ply. The stress resultants are then given by;

$$\{N_c\} = \sum_{k=1}^{n} \int_{h_{k-1}}^{h_k} \{\sigma_{\ell s}\}_k \, dz$$

$$\{M_c\} = \sum_{k=1}^{n} \int_{h_{k-1}}^{h_k} \{\sigma_{\ell s}\}_k \, z \, dz$$

These equations can be rewritten as

$$\{N_c\} = \sum_{k=1}^{n} \left([E_{\ell s}]_k \{\varepsilon_c^0\} \int_{h_{k-1}}^{h_k} dz + [E_{\ell s}]_k \{\kappa_c\} \int_{h_{k-1}}^{h_k} z \, dz \right)$$

$$\{M_c\} = \sum_{k=1}^{n} \left([E_{\ell s}]_k \{\varepsilon_c^0\} \int_{h_{k-1}}^{h_k} z \, dz + [E_{\ell s}]_k \{\kappa_c\} \int_{h_{k-1}}^{h_k} z^2 \, dz \right)$$

These equations can be simplified to

$$\{N_c\} = [A_c]\ \{\varepsilon_c^0\} + [C_c]\ \{\kappa_c\}$$

$$\{M_c\} = [C_c]\ \{\varepsilon_c^0\} + [D_c]\ \{\kappa_c\}$$

where,

$$[A_c] = \sum_{k=1}^{n} [E_{\ell s}]_k\ (z_k - z_{k-1})$$

$$[C_c] = \frac{1}{2} \sum_{k=1}^{n} [E_{\ell s}]_k\ (z_k^2 - z_{k-1}^2)$$

$$[D_c] = \frac{1}{3} \sum_{k=1}^{n} [E_{\ell s}]_k\ (z_k^3 - z_{k-1}^3)$$

Although the through-the-thickness stresses (σ_{zz}, σ_{yz}, σ_{xz}) are neglected in deriving the constitutive relationships for the orthotropic lamina, these stresses generally will exist in the laminate. These stresses are determined from equilibrium considerations and can be written as

$$Q_x = \frac{\partial M_x}{\partial x} - \frac{\partial M_{xy}}{\partial y}$$

$$Q_y = \frac{\partial M_{xy}}{\partial x} + \frac{\partial M_y}{\partial y}$$

$$\frac{\partial \sigma_z}{\partial z} = \frac{\partial \sigma_{xz}}{\partial x} + \frac{\partial \sigma_{yz}}{\partial y}$$

Assuming a parabolic distribution of shear stress (σ_{xz}, σ_{yz}) over the plate thickness, the shear stress for the ply can be written as

$$\sigma_{\ell xz} = \frac{3}{4}\,\frac{Q_x}{h}\left[1 - (\frac{z_\ell^0}{h})^2\right]$$

$$\sigma_{\ell yz} = \frac{3}{4}\,\frac{Q_y}{h}\left[1 - (\frac{z_\ell^0}{h})^2\right]$$

The through-the-thickness normal stress for the ply can be written as

$$\sigma_{\ell zz} = (\frac{\partial Q_x}{\partial_x} + \frac{\partial Q_y}{\partial_y})\ (\frac{3}{4}[\frac{z_\ell^0}{h} - \frac{1}{3}(\frac{z_\ell^0}{h})^3])$$

APPENDIX 1.3 - MATERIAL MODEL AND FIBER DEGRADATION

A modular multifactor constitutive model accounting for the effect on constituent properties of several parameters such as temperature, stress, and stress rate is employed. This model is shown below.

(1) Mechanical property (moduli, strength) P_M

$$\frac{P_M}{P_{Mo}} = \left[\frac{T_M - T}{T_M - T_o}\right]^n \left[\frac{S_F - \sigma}{S_F - \sigma_o}\right]^m \left[\frac{S_F - \dot{\sigma}_o}{S_F - \dot{\sigma}}\right]^l$$

(2) Thermal property (expansion coefficients, thermal conductivity, heat capacity) P_T

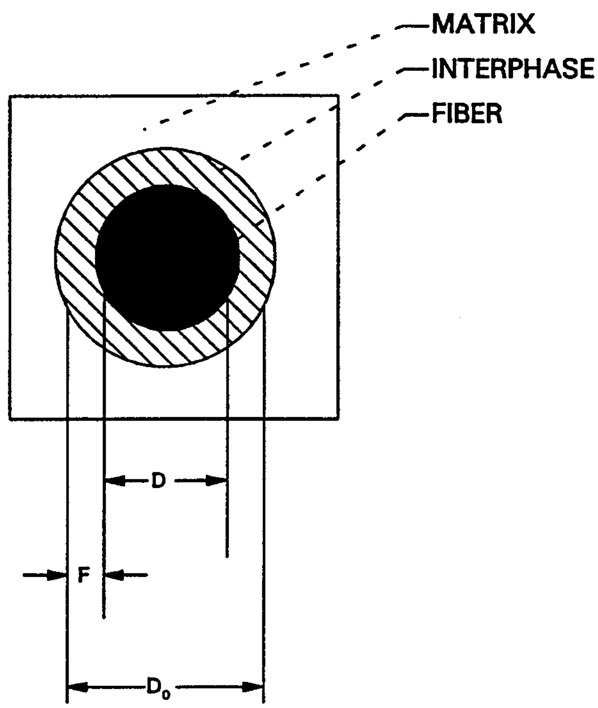
$$\frac{P_T}{P_{To}} = \left[\frac{T_M - T_o}{T_M - T}\right]^n \left[\frac{S_F - \sigma_o}{S_F - \sigma}\right]^m \left[\frac{S_F - \dot{\sigma}}{S_F - \dot{\sigma}_o}\right]^l$$

where

P_M	denotes the current mechanical property of interest
P_T	denotes the current thermal property of interest
P_{Me} , P_{To}	are corresponding properties at reference conditions
T_M	is the melting temperature
T	is the current temperature
T_o	is the reference temperature at which P_{Me} & P_{To} are determined
S_F	is the fracture stress determined at T_o conditions
σ	is the current stress
σ_o	is the reference stress at which P_{Me} & P_{To} are determined
$\dot{S}_F$	is an appropriately selected stress rate, for example, the stress rate at which penetration occurs during impact
$\dot{\sigma}_o$	is the stress rate at which P_{Me} & P_{To} are determined
$\dot{\sigma}$	is the stress rate, and
n, m, l	are empirical constants

Fiber degradation is a metallurgical phenomenon common in the refractory-fiber metal-matrix class of high temperature composites.
The phenomenon involves a chemical reaction at the fiber-matrix interface of a composite lamina. The exact nature and extent of the degradation process is dependant on the particular combination of fiber and matrix in the composite system. The result of this degradation process is the creation of an interphase of material at the original fiber/matrix boundary.

Depending on the propensity for a particular fiber/matrix system to interact and the character of the resulting interphase of material, the fiber degradation can have a significant deteriorating effect on the properties of the composite. Figure A1.6 illustrates the fiber degradation process used in METCAN. In the expression: D_o represents the initial fiber diameter, D the fiber diameter after degradation, and F is the percent of fiber degradation.



$$F = \frac{D_0 - D}{D_0}$$

FIBER DEGRADATION IN METCAN

FIGURE A1.6

MAT/FIB/DEGRAD3

APPENDIX 2

MHOST THEORY

As shown in Figure 2.3, the MHOST computer program is used to perform the global structural analysis. MHOST is a solid and structural analysis program based on mixed finite element technology, and is specifically designed for three-dimensional inelastic analysis. MHOST includes two- and three-dimensional elements along with beam and shell structural elements. Many options are available in the constitutive equation library, the solution algorithms, and the analysis capabilities. The options chosen in MHOST for use in HITCAN are:

- material option composite laminate for use with the plate element;

- material option anisotropic elasticity for the plane stress, plane strain, and eight-node solid elements;

- linear static, buckling, and modal analysis for the analysis option;

- displacement option for the finite element formulation.

In the following sections, brief theoretical reviews are given of those aspects of MHOST used in HITCAN. For further information the reader is directed to References 13, 14, and 15.

APPENDIX 2.1 - DISPLACEMENT FORMULATION

Using the displacement based formulation, the finite element problem can be reduced to the matrix equation

$$[K]\{u\} = \{F\},$$

where

$$[K] = \int_{\Omega}[B]^T[D][B]d\Omega$$

and where [B] and [D] are, respectively, the strain-displacement operator and the stress-strain relation for the element.

A solution is obtained by solving for the vector $\{u\}$ of unknown nodal displacements. Strains and stresses are not solved for directly. In MHOST, the strains are determined using

$$\{\epsilon_e\} = [B]\{u_e\},$$

where $\{\epsilon_e\}$is the strain defined at the integration points within the element and $\{u_e\}$ is the element nodal displacement vector. These strains are then extrapolated to the nodes. Here the stresses are calculated using

$$\{\sigma\} = [D]\{\epsilon\},$$

where $\{\sigma\}$ is the nodal stress vector, [D] is the stress-strain relation, and $\{\epsilon\}$ is the nodal strain vector.

APPENDIX 2.2 - ELEMENTS

The element library includes 3 four-node elements; i. e., plate, plane stress, and plane strain, and 1 eight-node element, a 3D solid. This section gives the theoretical background for these elements.

The plane stress element is a bilinear isoparametric element, based on selective reduced integration techniques. The element is a four noded quadrilateral with nodes numbered in a counter-clockwise fashion as shown in Figure A2.1. The position of each node is defined by a {x, y} coordinate pair.

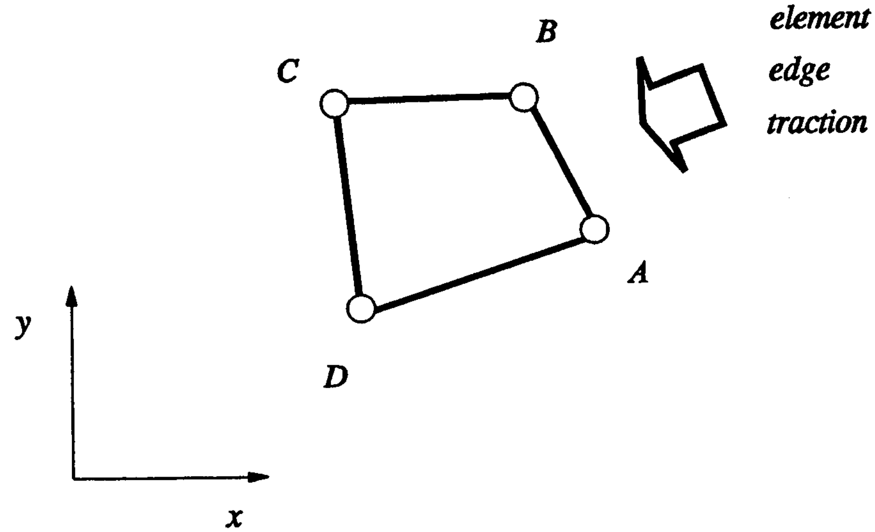
The displacements, strains, and stresses are stored as a set of vectors with components

$$\mathbf{u} = \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} \qquad \varepsilon = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \gamma_{xy} \end{Bmatrix} \qquad \sigma = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix}$$

where u_x and u_y are the two components of displacement, ϵ_x and ϵ_y are the two direct strain components, γ_{xy} is the shear strain (using the engineering definition), σ_x and σ_y are the direct stress components, and τ_{xy} is the in-plane stress.

The strain-displacement relation at a point x_i within the element is defined by

$$\varepsilon_i = \mathbf{B}_i \mathbf{u}^e \equiv \sum_{a=A}^{D} \mathbf{B}_{ia} \mathbf{u}_a$$



PLANE STRESS ELEMENT
FIGURE A2.1

where the $\{u_a\}$ for a = A, B, C, and D are the nodal displacement vectors. The [B] matrix can be partitioned into two submatrices

$$B_{ia} = \begin{bmatrix} B^d_{ia} \\ B^s_{ia} \end{bmatrix}$$

where $[B^d_{ia}]$ are the terms associated with the direct strain components

$$B^d_{ia} = \begin{bmatrix} \dfrac{\partial N_a}{\partial_x} & 0 \\ 0 & \partial\dfrac{N_a}{\partial_y} \end{bmatrix} x \equiv x_i$$

and $[B^s_{ia}]$ is used to compute the shear strain component

$$B^s_{ia} = \begin{bmatrix} \dfrac{\partial N_a}{\partial_y} & \dfrac{\partial N_a}{\partial_x} \end{bmatrix} x = x_i$$

The stress-strain relation used is

$$[D] = \begin{bmatrix} \dfrac{E_{11}}{(1-\mu_{21}\mu_{12})} & \dfrac{E_{11}\mu_{12}}{(1-\mu_{21}\mu_{12})} & 0 \\ \dfrac{E_{11}\mu_{12}}{(1-\mu_{21}\mu_{12}} & \dfrac{E_{22}}{(1-\mu_{21}\mu_{12})} & 0 \\ 0 & 0 & G_{12} \end{bmatrix}$$

The plane strain element is a bilinear isoparametric element. The element is a four-node quadrilateral with nodes numbered in a counter-clockwise fashion as shown in Figure A2.2.

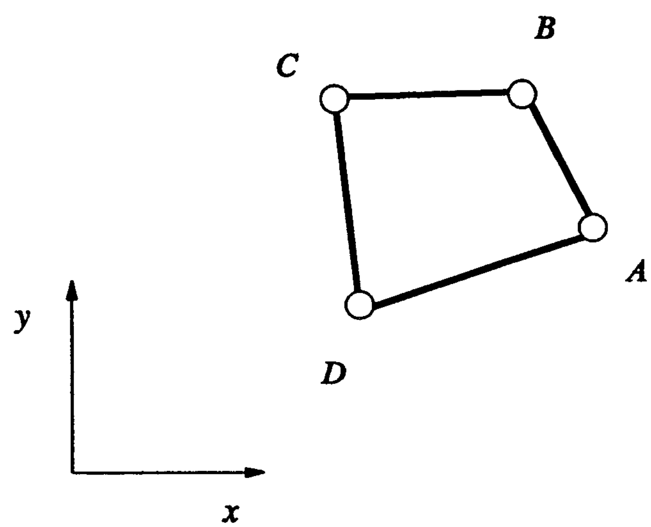
The displacements, strains, an stresses for this element are stored as a set of vectors with components

$$\mathbf{u} = \begin{Bmatrix} u_x \\ u_y \end{Bmatrix} \qquad \varepsilon = \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \end{Bmatrix} \qquad \sigma = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \end{Bmatrix}$$

where u_x and u_y are the two components of displacement, ϵ_x, ϵ_y, and ϵ_z are the direct strain components (with $\epsilon_z \equiv 0$), γ_{xy} is the shear strain (using the engineering definition), σ_x, σ_y, and σ_z are the direct stress components, and τ_{xy} is in-plane shear stress.

The strain-displacement relation at a point x_i within the element is defined by

$$\varepsilon_i = \mathrm{B}_i \mathrm{u}^e \equiv \sum_{a=A}^{D} \mathrm{B}_{ia} \mathrm{u}_a$$



PLANE STRESS ELEMENT
FIGURE A2.2

where the $\{u_a\}$ for a = A, B, C, and D are the nodal displacement vectors. The [B] matrix for the element can be partitioned into two submatrices

$$\mathbf{B}_{ia} = \begin{bmatrix} \mathbf{B}_{ia}^{d} \\ \mathbf{B}_{ia}^{s} \end{bmatrix}$$

where $[B^d_{ia}]$ are the terms associated with the direct strain components

$$\mathbf{B}_{ia}^{d} = \begin{bmatrix} \dfrac{\partial N_a}{\partial x} & 0 \\ 0 & \dfrac{\partial N_a}{\partial y} \\ 0 & 0 \end{bmatrix}_{x \equiv x_i}$$

whereas $[B^s_{ia}]$ is used to compute the shear strain component.

$$\mathbf{B}_{ia}^{s} = \begin{bmatrix} \dfrac{\partial N_a}{\partial y} & \dfrac{\partial N_a}{\partial x} \end{bmatrix}_{x \equiv x_i}$$

The stress-strain relation used is

$$\begin{bmatrix} \frac{E_{11}}{C_1} & \frac{(\mu_{21}+\mu_{31}\mu_{23}E_{11})}{C_1(1-\mu_{23}\mu_{32})} & 0 & 0 \\ \frac{E_{22}(1-\mu_{31}\mu_{31})}{C_2(1-\mu_{31}\mu_{32})} & \frac{E_{22}}{C_2} & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & G_{12} \end{bmatrix}$$

where

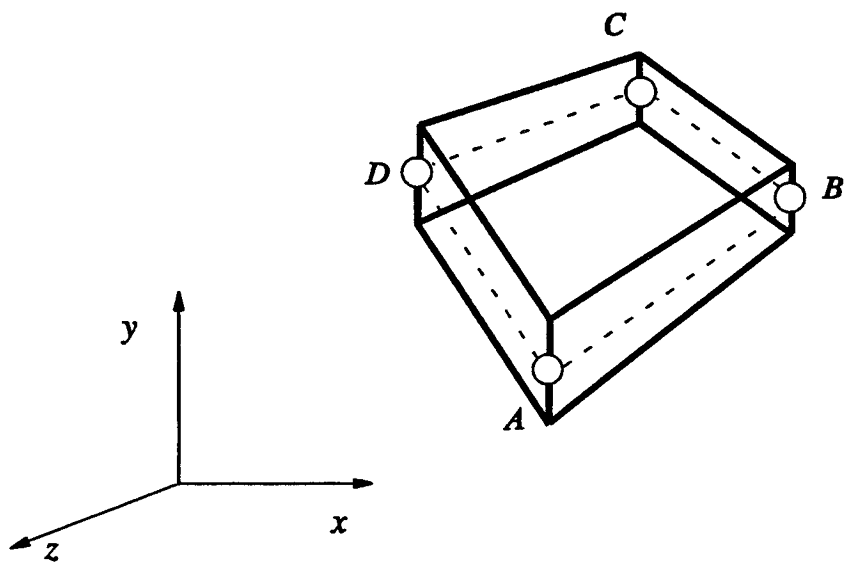
$$C_1=(1-\mu_{13}\mu_{31})- \ [(\mu_{21}+\mu_{31}\mu_{23}) \ (\mu_{12}+\mu_{13}\mu_{32})/(1-\mu_{23}\mu_{32})]$$
$$C_2=(1-\mu_{23}\mu_{32})- \ [(\mu_{21}+\mu_{31}\mu_{23}) \ (\mu_{12}+\mu_{13}\mu_{32})/(1-\mu_{13}\mu_{31})]$$

The plate element is a bilinear isoparametric variable-thickness plate element based on the Reissner-Mindlin plate and shell theories. These theories account for shear deformation, and can be applied to moderately thick shells.

The element is a four-noded quadrilateral in three-dimensional space as shown in Figure A2.3. Each node is defined by four coordinates {x, y, z, t} where t denotes the thickness of the shell at each node.

The plate element is formulated in terms of global translations and rotations, generalized strains, and stress resultants. These displacements, generalized strains and stress resultants are stored as a set of vectors with the components

$$u = \begin{Bmatrix} u_x \\ u_y \\ u_z \\ \theta_x \\ \theta_y \\ \theta_z \end{Bmatrix} \qquad \varepsilon = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \\ \psi_{xy} \end{Bmatrix} \qquad \sigma = \begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ S_{xz} \\ S_{yz} \\ M_x \\ M_y \\ M_{xy} \\ \Psi_{xy} \end{Bmatrix}$$



PLANE ELEMENT

FIGURE A2.3

where u_x, u_y, and u_z are the three translational displacements, Θ_x, Θ_y, and Θ_z represent three rotations, all expressed in the global Cartesian coordinate system; ϵ_x and ϵ_y are two in-plane strains, γ_{xy} is in-plane shear strain, γ_{yz}, and γ_{zx} are the transverse shear strains, κ_x, κ_y, and κ_z denote the three curvatures at a point, and Ψ_z is the term associated with the "drilling" rotation, all expressed in the local element coordinate system; N_x and N_y are the in-plane membrane forces, N_{xy} the in-plane shear, S_{xz} and S_{yz} are two transverse shears, M_x, M_y, and M_{xy} are the bending moments at a point, and S_{xy} the force resultant associated with the "drilling" mode, all expressed in the local element coordinate system.

As mentioned previously, the plate element is a bilinear isoparametric variable-thickness element based on the Reissner-Mindlin plate and shell theories. The weak variational form for these equations involves only first-order derivatives of displacement, permitting the use of linear interpolation for both rotations and transverse displacements. The strain-displacement relation for an arbitrary point x_i at the mid-surface of the plate is defined by

$$\epsilon_i = B_i u^e \equiv \sum_{a=A}^{D} B_{ia} u_a$$

where the $\{u_a\}$ for a = A, B, C, and D are the nodal displacement vectors. The [B] matrix for the element can be partitioned into four submatrices

$$B_{ia} = \begin{bmatrix} B_{ia}^{m} \\ B_{ia}^{s} \\ B_{ia}^{b} \\ B_{ia}^{d} \end{bmatrix}$$

$$B_{ia}^{m}=\begin{bmatrix} \frac{\partial N_a}{\partial_x} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{\partial N_a}{\partial_y} & 0 & 0 & 0 & 0 \\ \frac{\partial N_a}{\partial_y} & \frac{\partial N_a}{\partial_x} & 0 & 0 & 0 & 0 \end{bmatrix}_{x=x_i}$$

where $[B^m_{ia}]$ denotes the terms associated with the two direct membrane strain components

$$B_{ia}^{s}=\begin{bmatrix} 0 & 0 & \frac{\partial N_a}{\partial_x} & 0 & N_a & 0 \\ 0 & 0 & \frac{\partial N_a}{\partial_y} & -N_a & 0 & 0 \end{bmatrix}_{x=x_i}$$

where $[B^s_{ia}]$ denotes the terms associated with the two direct transverse shear strain components

$$B_{ia}^{b}=\begin{bmatrix} 0 & 0 & 0 & 0 & -\frac{\partial N_a}{\partial_x} & 0 \\ 0 & 0 & 0 & \frac{\partial N_a}{\partial_y} & 0 & 0 \\ 0 & 0 & 0 & \frac{\partial N_a}{\partial_x} & -\frac{\partial N_a}{\partial_y} & 0 \end{bmatrix}_{x=x_i}$$

where $[B^b_{ia}]$ denotes the terms associated with the two direct bending strain components

$$B_{ia}^{d}=\begin{bmatrix} -\frac{\partial N_a}{\partial_y} & \frac{\partial N_a}{\partial_x} & 0 & 0 & 0 & -2N_a \end{bmatrix}_{x=x_i}$$

where $[B^d_{ia}]$ denotes the terms associated with the two direct drilling strain components

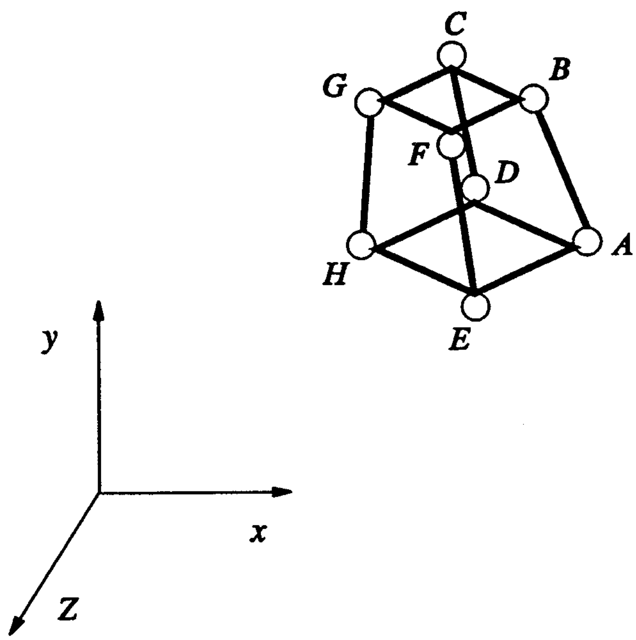
The stress-strain relation for this element is entered into MHOST using the axial, bending, and coupling matrices of laminate theory.

The eight-node solid element is a trilinear isoparametric element. The element is shown in Figure A2.4. The position of each node is defined by a {x, y, z} coordinate vector.

The displacements, strains, and stresses of this element are stored as a set of vectors with components

$$\mathbf{u}=\begin{Bmatrix} u_x \\ u_y \\ u_z \end{Bmatrix} \qquad \boldsymbol{\varepsilon}=\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \qquad \boldsymbol{\sigma}=\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$$

where u_x, u_y, and u_z are the three components of displacement, ϵ_x, ϵ_y, and ϵ_z are the direct strain components, γ_{xy}, γ_{yz}, and γ_{zx}
are the three shear strains (using the engineering definition), σ_x, σ_y, and σ_z are the direct stress components, and τ_{xy}, τ_{yz}, and τ_{zx}
are the three shear stresses.



EIGHT-NODE SOLID ELEMENT

FIGURE A2.4

The strain-displacement relation at a point x_i within the element is defined by

$$\varepsilon_i = B_i u^e \equiv \sum_{a=A}^{H} B_{ia} u_a$$

where the $\{u_a\}$ for a = A, B, ..., H are the nodal displacement vectors. The [B] matrix for the element can be partitioned into two submatrices

$$B_{ia} = \begin{bmatrix} B_{ia}^{d} \\ B_{ia}^{s} \end{bmatrix}$$

where $[B^d_{ia}]$ are the terms associated with the direct strain components

$$B_{ia}^{d} = \begin{bmatrix} \frac{\partial N_a}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_a}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_a}{\partial z} \end{bmatrix}_{x \equiv x_i}$$

whereas $[B^s_{ia}]$ is used to compute the shear strain components

$$B_{ia}^{s} = \begin{bmatrix} \frac{\partial N_a}{\partial y} & \frac{\partial N_a}{\partial x} & 0 \\ \frac{\partial N_a}{\partial z} & 0 & \frac{\partial N_a}{\partial x} \\ 0 & \frac{\partial N_a}{\partial z} & \frac{\partial N_a}{\partial y} \end{bmatrix}_{x \equiv x_i}$$

The stress-strain relation used is

$$[D] = \begin{bmatrix} 1-\dfrac{\mu_{23}\mu_{32}}{CE_{22}E_{33}} & \dfrac{\mu_{12}-\mu_{32}\mu_{13}}{CE_{11}E_{33}} & \dfrac{\mu_{13}-\mu_{12}\mu_{23}}{CE_{11}E_{22}} & 0 & 0 & 0 \\ \dfrac{\mu_{12}-\mu_{32}\mu_{13}}{CE_{11}E_{12}} & \dfrac{1-\mu_{13}\mu_{31}}{CE_{11}E_{33}} & \dfrac{\mu_{23}-\mu_{21}\mu_{13}}{CE_{22}E_{33}} & 0 & 0 & 0 \\ \dfrac{\mu_{13}-\mu_{12}\mu_{13}}{CE_{11}E_{22}} & \dfrac{\mu_{23}-\mu_{21}\mu_{13}}{CE_{22}E_{33}} & \dfrac{1-\mu_{12}\mu_{21}}{CE_{11}E_{22}} & 0 & 0 & 0 \\ 0 & 0 & 0 & G_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & G_{23} & 0 \\ 0 & 0 & 0 & 0 & 0 & G_{31} \end{bmatrix}$$

$$where \quad C = \frac{1-\mu_{12}\mu_{21}-\mu_{23}\mu_{32}-\mu_{31}\mu_{13}-2\mu_{21}\mu_{32}\mu_{13}}{E_{11}E_{22}E_{33}}$$

APPENDIX 2.3 - BUCKLING AND MODAL ANALYSIS

In this section, a brief review of the theoretical aspects of buckling analysis and modal analysis is given. In MHOST both of these analyses are formulated as eigenvalue problems. The subspace iteration method is used in MHOST to determine the eigenvalues and eigenvectors.

The eigenvalue problem associated with the critical buckling load can be written as

$$([K] + \lambda[K_\sigma])\{u\} = \{0\},$$

where $[K_\sigma]$ is the stress stiffness matrix, $[K]$ is the linear stiffness matrix, and λ some scaler multiplier of $[K_\sigma]$. Since MHOST, as it is used in HITCAN, only performs a linear analysis, then neither $[K]$ or $[K_\sigma]$ are functions of displacement. The critical buckling load is associated with λ_{cr}, the lowest magnitude eigenvalue. The displacement vector $\{u\}$ identifies the buckled shape, but not its magnitude. For further information, see Reference 12.

The eigenvalue problem associated with modal analysis is

$$([K] - \lambda[M])\{u\} = \{0\},$$
$$\lambda = \omega^2,$$

where $[K]$ is the stiffness matrix, $[M]$ is the mass matrix, $\{u\}$ is the amplitude of the nodal degrees-of-freedom, ω is the frequency, and λ is the eigenvalue. Since neither geometric nor material nonlinearity are used in MHOST, then neither $[K]$ or $[M]$ is a function of ω. Only certain values λ_i will satisfy the above equations. These λ_i are called eigenvalues. To each eigenvalue there corresponds an eigenvector $\{u_i\}$. The eigenvalue problem is to extract the solution pairs λ_i and $\{u_i\}$.

As mentioned previously, the subspace iteration method is used to solve the eigenvalue problem. For a description of this method, see Reference 14.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY *(Leave blank)*

2. REPORT DATE
April 1993

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
High Temperature Composite Analyzer (HITCAN) Theoretical Manual
Version 1.0

5. FUNDING NUMBERS
WU-505-63-5B

6. AUTHOR(S)
J.J. Lackney, P.L.N. Murthy, and P.K. Gotsis

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135-3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E-7544

9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, D.C. 20546-0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM-106001

11. SUPPLEMENTARY NOTES
J.J. Lackney, Sverdrup Technology, Inc., Lewis Research Center Group, 2001 Aerospace Parkway, Brook Park, Ohio 44142 (work funded by NASA Contract NAS3-25266); and P.L.N. Murthy and P.K. Gotsis, Lewis Research Center, Cleveland, Ohio. Responsible person, P.L.N. Murthy, (216) 433-3332.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category 39

12b. DISTRIBUTION CODE

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

This manual outlines some of the theoretical aspects embedded in the computer code, HITCAN (HIgh Temperature Composite ANalyzer). HITCAN is a general purpose computer program for predicting nonlinear global structural and local stress-strain response of arbitrarily oriented, multilayered high temperature metal matrix composite structures. This code combines composite mechanics and laminate theory with an internal data base of the constituents (matrix, fiber and interface) material properties. The thermal and mechanical properties of the constituents are considered to be nonlinearly dependent on several parameters including temperature, stress and stress rate. The computational procedure for the analysis of the composite structure uses the finite element method. HITCAN consists of three modules: COBSTRAN, METCAN and MHOST. COBSTRAN generates the geometry (pre-processor) and defines the lay up of the different plies. METCAN computes the material behavior of the composite and of the constituents. Finally, MHOST is a finite element program based on the mixed iterative solution technique. MHOST has a library for 2D and 3D isoparametric elements. HITCAN is written in FORTRAN 77 computer language and has been configured and executed on the NASA Lewis Research Center CRAY XMP and YMP computers.

14. SUBJECT TERMS
Metal matrix composite structures; Fiber composites; Composite mechanics; Laminate analysis; Microstresses; Micromechanics; High temperature; Thermal properties; Material nonlinearities; 2D and 3D nonlinear analysis; Finite elements; Buckling; Stress analysis; Vibration frequencies

15. NUMBER OF PAGES
58

16. PRICE CODE
A04

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

PRECEDING PAGE BLANK NOT FILMED