NASA-CR-189,089

NASA Contractor Report 189089

3D Inelastic Analysis Methods for Hot Section Components

Final Report

R.L McKnight, R.J. Maffeo,
M.T. Tipton, and G. Weber

General Electric Company
Aircraft Engines
Cincinnati, Ohio 45215

January 1992

Prepared For
Lewis Research Center
Under Contract NAS3-23698

National Aeronautics and
Space Administration

3D Inelastic Analysis Methods for Hot Section Components

Final Report

January 1992

National Aeronautics and
Space Administration

Table of Contents

List of Illustrations

Figure | Page

List of Tables

NOMENCLATURE

A, m, R	-	Material Constants for Bodner's Model
C_{ijmn}	-	Effective Modulus Tensor
D_{ijmn}	-	Elastic Constitutive Tensor
D_o	-	Limiting Shear Strain Rate
dP_{ij}	-	Plasticity Correction Tensor
$\{d^E\}$	-	Elastic Displacement Vector
$\{d^I\}$	-	Inelastic Displacement Vector
$\{d^T\}$	-	Total Displacement Vector
E	-	Modulus of Elasticity
E_{mn}	-	Green-Lagrange Strain Tensor
E^c_{mn}	-	Creep Strain Tensor
E^p_{mn}	-	Plastic Strain Tensor
E^T_{mn}	-	Thermal Strain Tensor
e	-	Maximum Percent Error
H^1	-	Slope of Uniaxial Stress-Strain Curve
J_2	-	Second Invariant of the Deviatoric Stress
K	-	Yield Stress Radius
N_i	-	Isoparametric Shape Functions
n	-	Bodner's Strain Rate Sensitivity Constant
Q,r	-	Creep Constants
S_{ij}	-	Deviatoric Stress Tensor

T	-	Temperature
V_{i1}, V_{i2}, V_{i3}	-	Orthogonal Local Triad
$\dot{W}^I$	-	Inelastic Work Rate
x, y, z	-	Cartesian Coordinates
Z	-	Bodner's State Variable
Z_1	-	Maximum Value of Z
Z_2	-	Minimum Value of Z
α_{ij}	-	Coordinates of Yield Surface Center
$\Delta\varepsilon^C$	-	Increment in Creep Strain
ε_e	-	Total Effective Strain
$\Delta\varepsilon_e$	-	Total Effective Strain Increment
ε_e^E	-	Effective Elastic Strain
$\Delta\varepsilon_e^E$	-	Effective Elastic Strain Increment
$\Delta\varepsilon_e^P$	-	Effective Plastic Strain Increment
ε_{ij}	-	Strain Tensor
$\dot{\varepsilon}_{ij}^I$	-	Inelastic Strain Rate Tensor
$d\varepsilon_{ij}^P$	-	Incremental Plasticity Tensor
$d\bar{\varepsilon}^P$	-	Equivalent Uniaxial Plastic Strain History
$d\gamma$	-	Prandtl-Reuss Scale Factor
$d\mu$	-	Yield Surface Translation Scalar
ν	-	Poisson's Ratio
ξ, η, ζ	-	Barycentric Coordinates
σ_e	-	Effective Stress
σ_{em}	-	Maximum Effective Stress
σ_{ij}	-	Stress Tensor
ψ	-	Yield Surface

1.0 INTRODUCTION

The objective of this research was to develop analytical tools capable of economically evaluating the cyclic time-dependent plasticity which occurs in hot section engine components in areas of strain concentration resulting from the combination of both mechanical and thermal stresses. The techniques developed are capable of accomodating large excursions in temperatures with the associated variations in material properties including plasticity and creep.

The overall objective of this research program was to develop advanced 3-D inelastic structural/stress analysis methods and solution strategies for more accurate and yet more cost-effective analysis of combustors, turbine blades, and vanes. The approach was to develop a matrix of formulation elements and constitutive models, three increasingly more complex formulation models and three increasingly more complex constitutive models.

The three constitutive models were developed in conjunction with optimized iteration techniques, accelerators, and convergence criteria within a framework of dynamic time incrementing. These consist of a simple model, a classical model, and a unified model. The simple model performs time-independent inelastic analyses using a bilinear stress-strain curve and time-dependent inelastic analyses using a power-law creep equation. The second model is the classical model of Professors Walter Haisler and David Allen (Reference 1) of Texas A&M University. The third model is the unified model of

Bodner, Partom, et al. (Reference 2). All models were customized for a linear variation of loads and temperatures with all material properties and constitutive models being temperature dependent.

The three formulation models developed are an eight-noded midsurface shell element, a nine-noded midsurface shell element and a twenty-noded isoparametric solid element. Both of the shell elements are obtained by "degenerating" 3D isoparametric solid elements and then imposing the necessary kinematic assumptions in connection with the small dimension of the shell thickness (References 3 and 4). The eight-noded element uses Serendipity shape functions and the nine-noded element uses Lagrange shape functions. The eight-noded element uses Gaussian quadrature for numerical integration, with nodal and surface stresses being obtained by extrapolation/mapping techniques. Lobatto quadrature is being used with the nine-noded element to effectively provide for direct recovery of the stresses and strains at the surfaces and node points. The eight-noded element has an excellent combination of accuracy and economy in the normal element aspect range encountered when modeling most hot section components. The nine-noded Lagrangian formulation overcomes the shear locking problem experienced when the element size-versus-thickness-aspect ratio becomes very large. The twenty-noded isoparametric element uses Gaussian quadrature.

A separate computer program has been developed for each combination of constitutive model-formulation model. Each program provides a functional, stand alone capability for performing cyclic nonlinear structural analysis. In addition, the analysis capabilities incorporated into each program can be abstracted in subroutine form for incorporation into other codes or to form new combinations. These programs will provide the structural analyst with a matrix of capabilities involving the constitutive models-formulation models from which he will be able to select the combination that satisfies his particualr needs.

The program architecture employs state-of-the-art techniques to maximize efficiency, ut...ity, and portability. Among these features are the following:

(i) User Friendly I/O

- Free format data input
- Global, local coordinate system, (Cartesian, Cylindrical, Spherical)
- Automatic generation of nodal and elemental attributes
- User-controlled optional print out

  Nodal Displacements

  Nodal Forces

  Element Forces

  Element Stresses and Strains

(ii) Programming Efficiency

- Dynamic core allocation
- Optimization of file/core utilization
- Blocked column skyline out-of-core equation solver

(iii) Accurate and Economical Solution Techniques

- Right-hand side pseudoforce technique
- Accelerators for the iteration scheme
- Convergence criteria based on both the local inelastic strain and the global displacements.

The ability to model piecewise linear load histories was also included in the finite element codes. Since the inelastic strain rate could be expected to change dramatically during a linear load history, it is important to include a dynamic time-incrementing procedure.

Three separate time step control criteria are used. These are the maximum stress increment, maximum inelastic strain increment, and maximum rate of change of the inelastic strain rate. The minimum time step calculated from the three criteria is the value actually used. Since the calculations are based on values readily available from the previous time step, little computational effort is required.

These formulation models and constitutive models have been checked out extensively against both theory and experiment. Figure 1 shows the correlation between Bodner's model in the eight-noded and mid-surface shell element (MSS8) and both experiment and other predictions (Reference 5). Figures 2 through 6 illustrate the predictability of the classical Hiasler-Allen model. Figure 7 shows a comparison of both Bodner's model and the simple model to both experiment and independent predictions (Reference 6).

These nine programs, both source (Fortran 77) and compiled, have been installed and checked out on the NASA-Lewis CRAY-1 machine. The interactive deck generator has been installed on the NASA-Lewis AMDAHL machine.

Table 1 shows the lines of source code for each of the nine computer programs. These numbers do not include the interactive deck generators.

Table 1. Lines of Source Code

		Elements		
		20-Noded	8-Noded	9-Noded
Constitutive Models	Simple	8300	13,800	17,900
	Haisler-Allen	9200	16,300	19,000
	Bodner	7300	13,800	17,600

Since these programs use dynamic core allocation, they can be recompiled to size for any specific machine. They are presently loaded for 10^7 bytes of core. At this size, the maximum problem would be approximately 4000 nodes and 1000 elements, and 24000 degrees of freedom.

REFERENCES

1. Allen, D.H., and Haisler, W.E., "A Theory of Thermoplastic Materials," Computers and Structures, Vol. 13, pp. 129-135, 1981.

2. Bodner, S.A., Partom, I., and Partom, Y., "Uniaxial Cyclic Loading of Elastic-Viscoplastic Material, "ASME J. Appl. Mech., Vol. 46, p. 805, 1979.

3. Chang, T.Y. and Sawamiphakdi, K., "Large Deformation Analysis of Shells by Finite Element Method," Comp. and Struct., Vol. 13, pp. 331-340, 1981.

4. Chang, T.Y. and Sawamiphkdi, K., "Nonlinear Finite-Element Analysis of Shells with Large Aspect Ratio," presented at the Nonlinear Structural Analysis Workshop, NASA-Lewis Research Center, April 19 and 20, 1983.

5. Stouffer, D.C., "A Constitutive Representation for IN100," Air Force Materials Laboratory, AFWAL-TR-81-4039,1981.

6 Bodner, S.R., "Representation of Time Dependent Mechanical Behavior of Rene'95 by Constitutive Equations," Air Force Materials Laboratory, AFML-TX-79-4116, 1979.

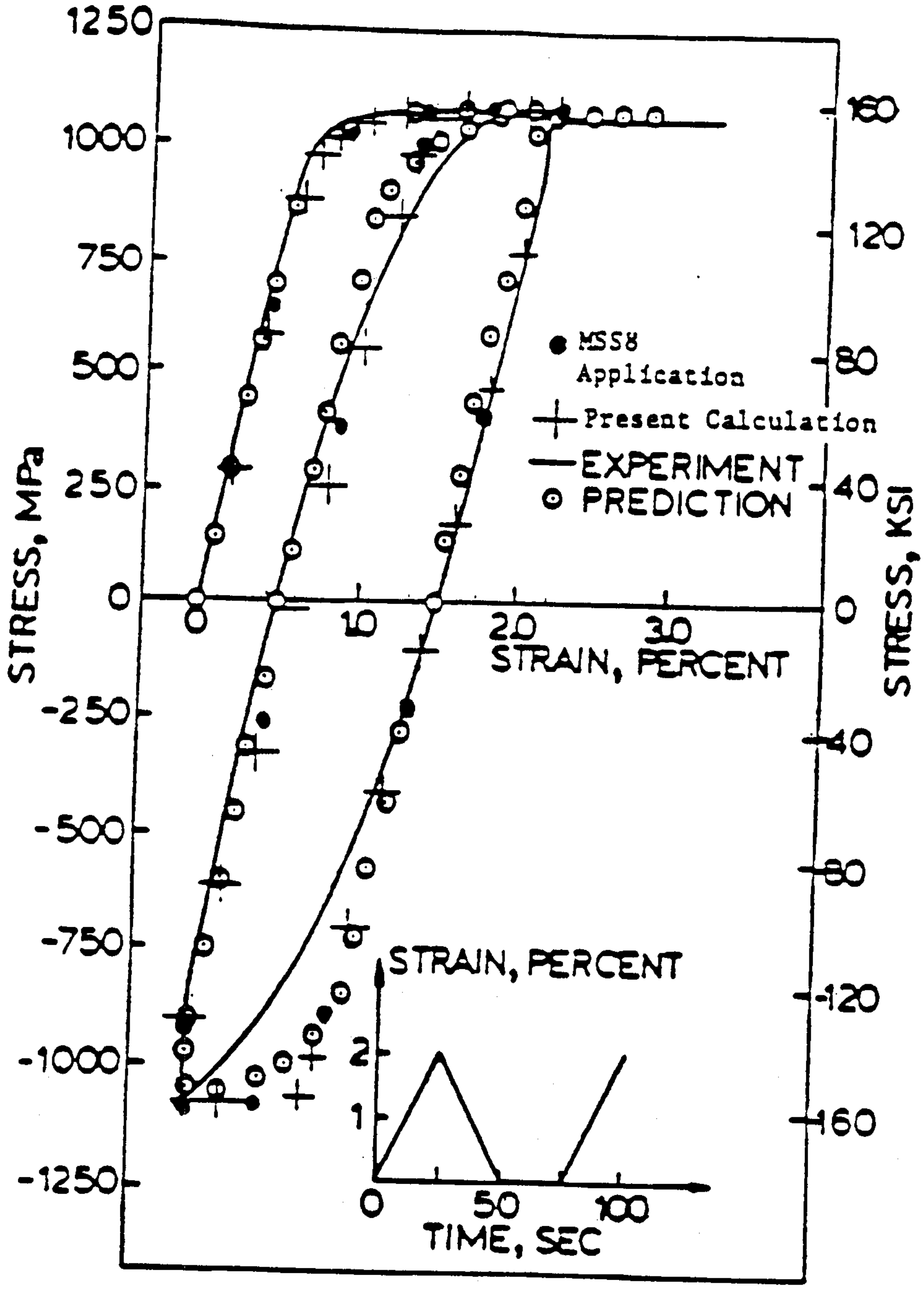


Figure 1. Displacement Controlled Cycling Results.

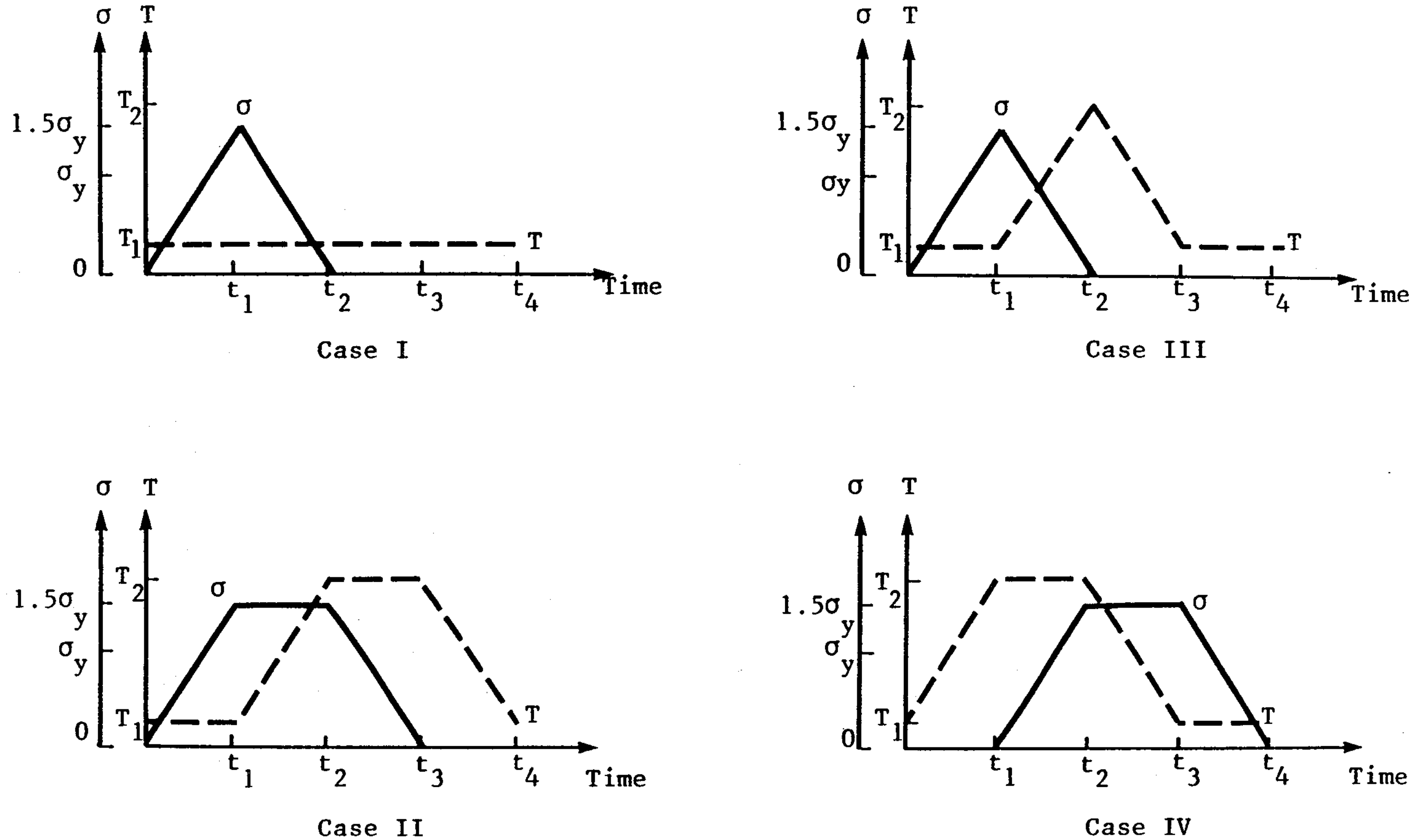


Figure 2. Load Histories for Plasticity Example.

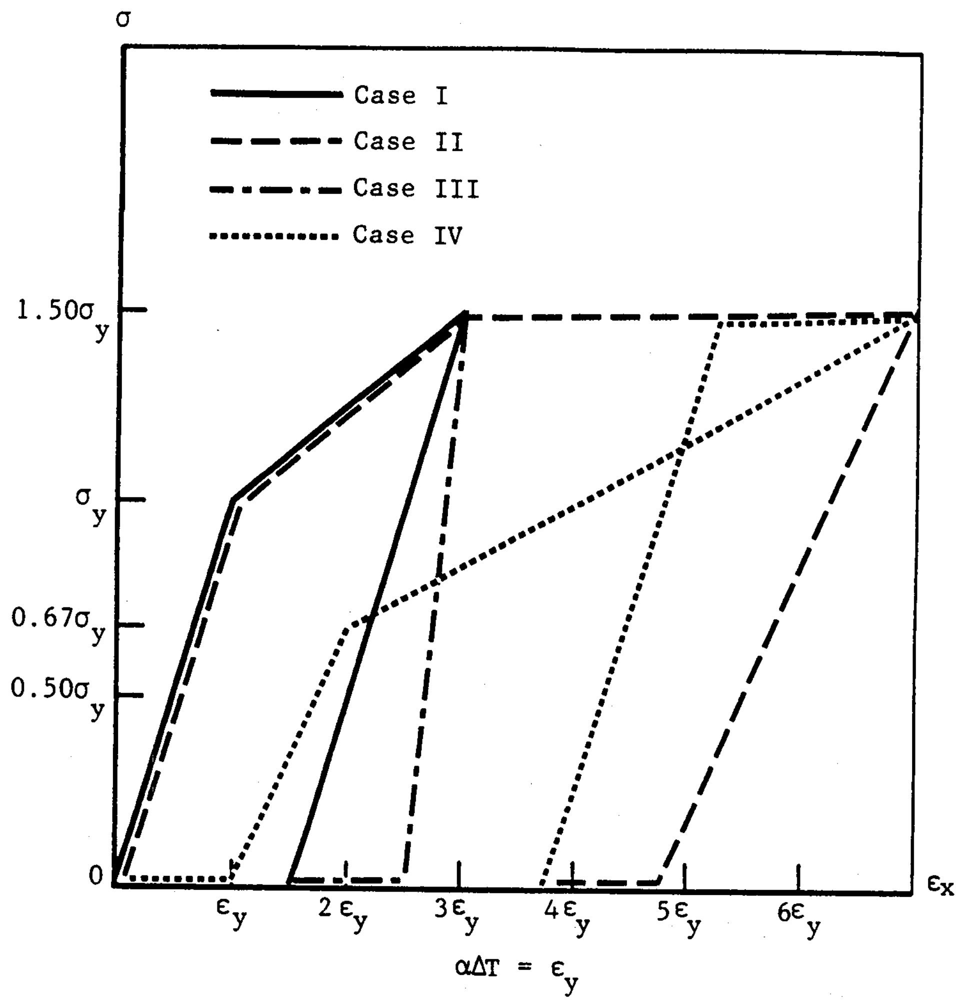


Figure 3. Results of Plasticity Example.

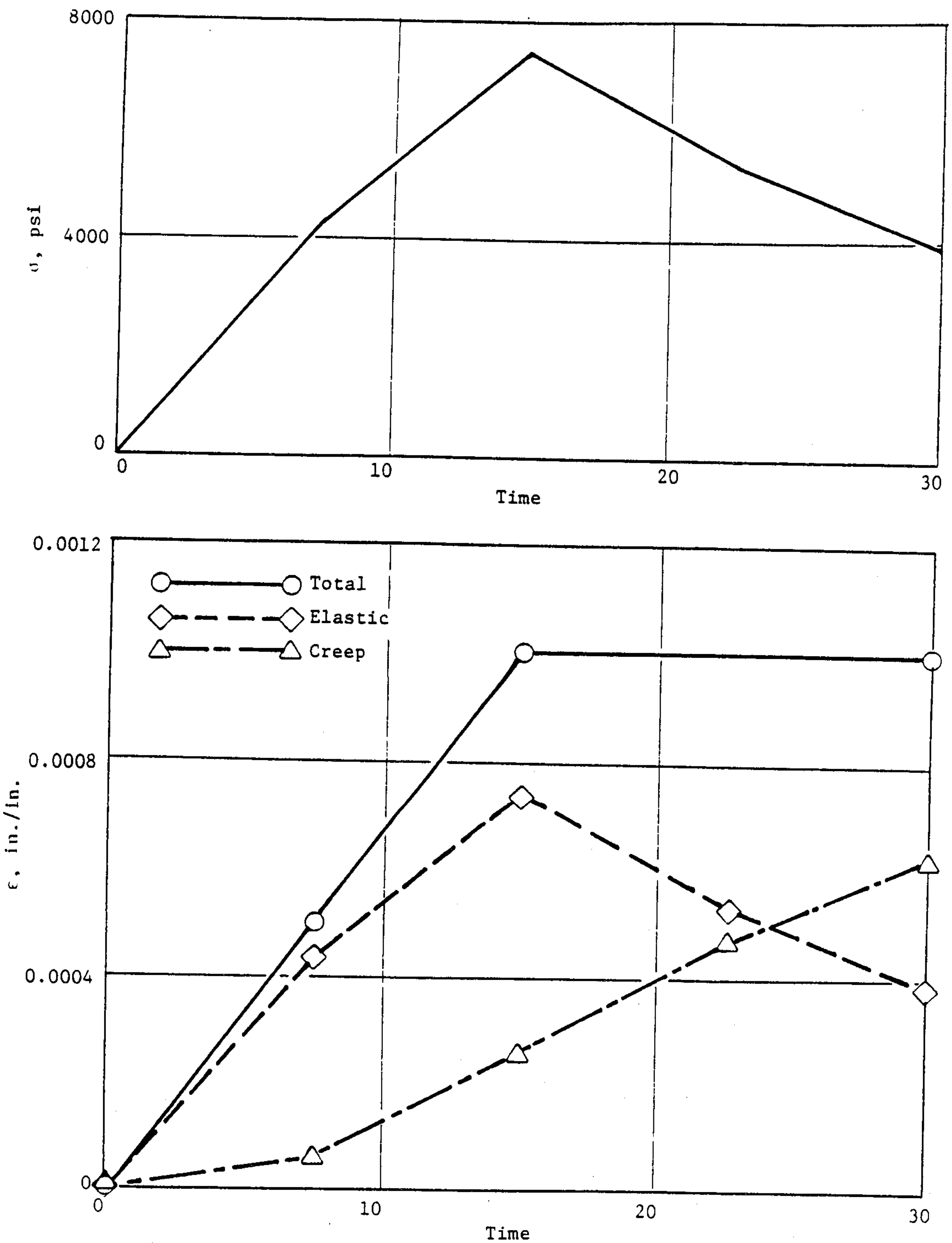


Figure 4. Strain Controlled Creep.

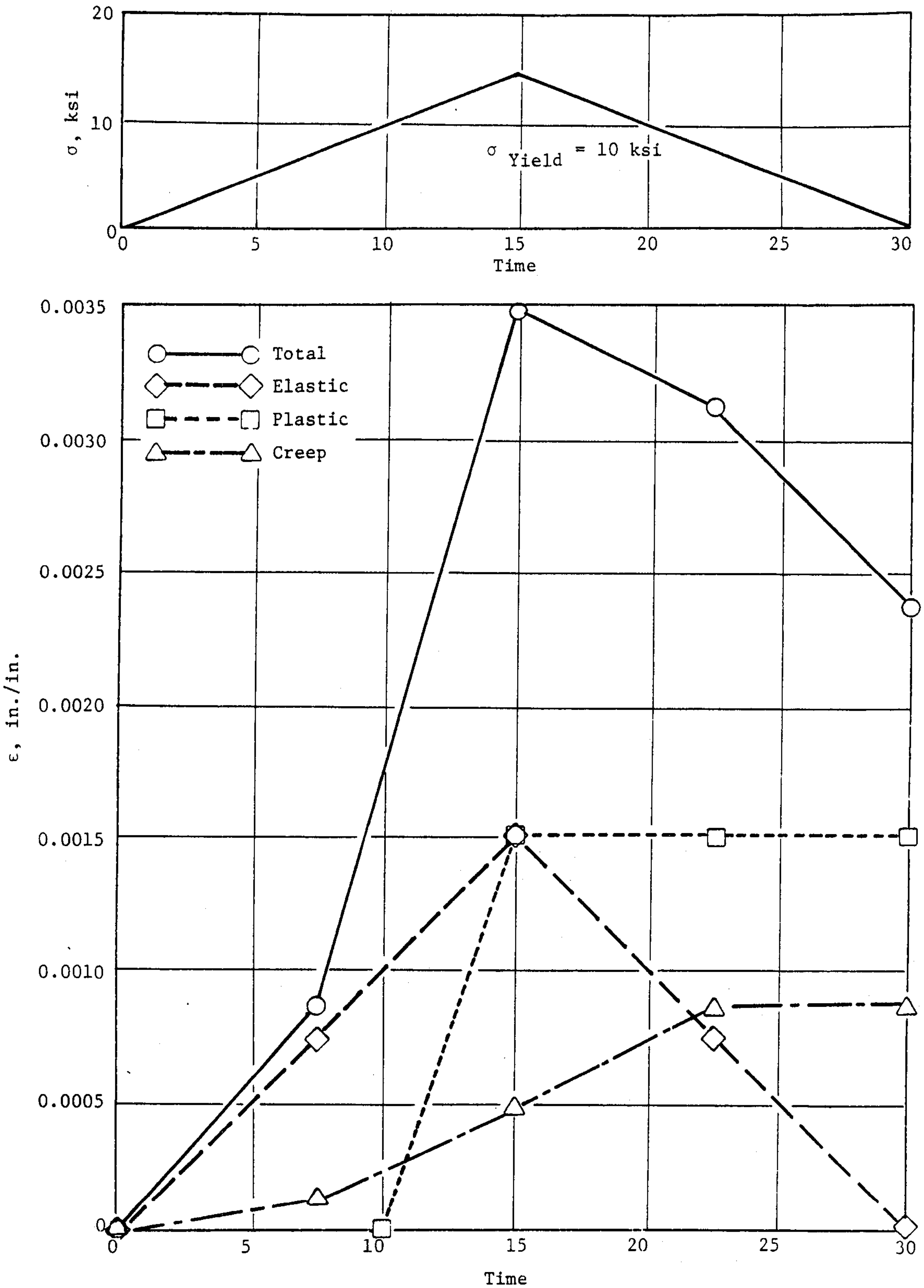


Figure 5. Stress Controlled Cycling with Combined Plasticity and Creep.

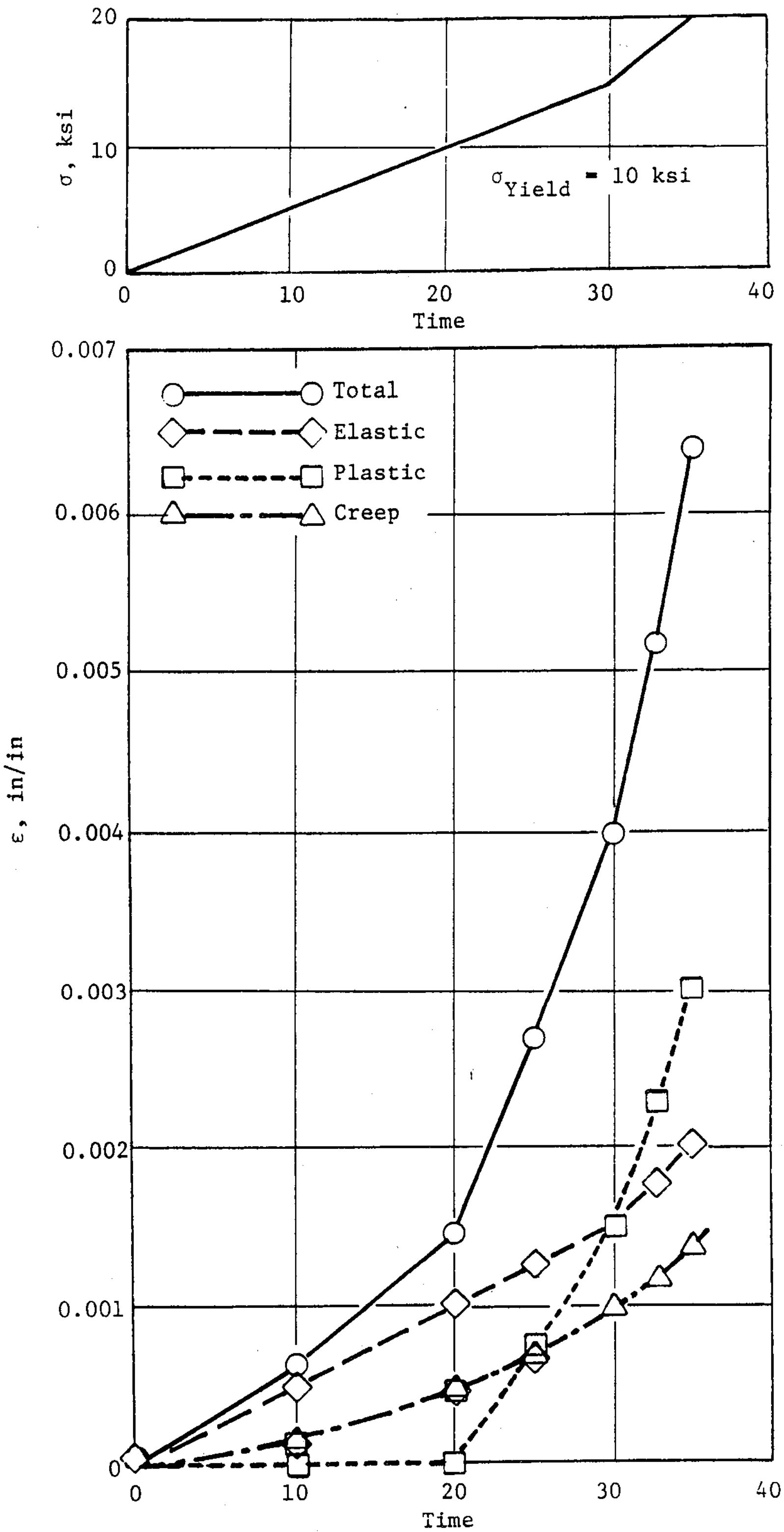


Figure 6. Stress Controlled Combined Plasticity and Creep.

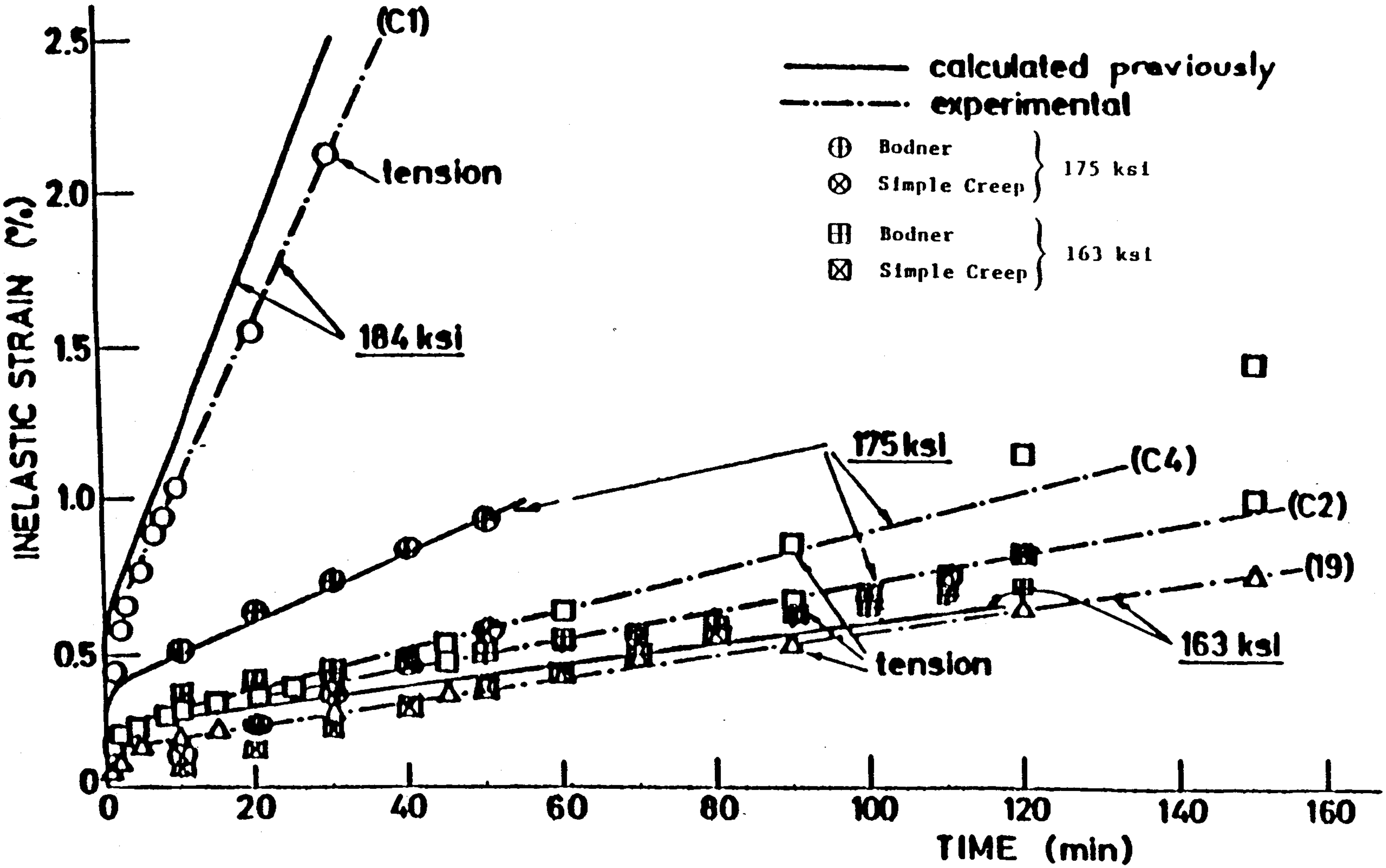


Figure 7. Comparison of Bodner Mode to Simple Creep.

2.0 TECHNICAL PROGRESS

The first activity in this program was the performance of a literature survey. The pertinent results of this are given in Appendix A. Based on the results of this survey, three constitutive models and three formulation models have been developed.

The final versions of the computer programs for the 3D Inelastic Analysis Methods contract have been installed on the NASA-Lewis CRAY. There are nine separate programs, each with a different combination of the 3 element types and 3 constitutive models. The element types are a 20-noded insoparametric brick element, an 8-noded midsurface shell element, and a 9-noded midsurface shell element. The constitutive models are a simple isotropic hardening classical plasticity model, a sophisticated classical plasticity model with combined kinematic and isotropic hardening, and a unified model. Both classical plasticity models also have a secondary creep model combined with them.

The major features of the programs are described below. These features are generally available in each of the programs except as specifically stated.

The physical geometry of the finite element models is described by nodal locations and element connectivities. The shell programs provide for automatic generation of nodes and elements if requested. Since quadratic shape functions are used in all the element types straight or curved boundaries that can be described by a quadratic are easily modeled simply by appropriately placing the nodes on the element edge around the boundary. No special information is necessary.

Loadings can be specified in the form of nodal concentrated loads, displacements, and/or temperatures or on an element level as pressures for the 20-noded brick element and as pressures and line loads in the shell program. Global loadings such as gravity and rotational loads can also be specified. Loads with the same value can be easily applied to large numbers of nodes and elements since series of evenly incremented node and element numbers can be specified.

All of the programs provide for linear ramping of loadings. Initial and final conditions for the load case are input and the intermediate conditions are obtained by linear interpolation. The intermediate points can be specified in a couple of ways. A number of even load increments can be specified with or without a reference to time. Also available is a dynamic time incrementing scheme which allows the program to calculate time increments and that user input maximums of stress change, inelastic strain change, or inelastic strain integration error are not exceeded.

In this way the program will use large time steps for load intervals causing small inelastic action and small time steps for loadings causing a great deal of inelastic action to occur.

The constitutive models are capable of predicting inelastic responses for constant or variable temperature conditions This is done by including terms that arise due to thermal variation in the derivation of these models and by allowing the user to specify the required inelastic data at up to 10 different temperatures per material type. There can be up to 3 different material types per model. Each of the constitutive models is capable of modeling response due to simple loading as well as reverse or cyclic loadings.

The elastic material properties (elastic modulus, Poisson's ratio, and thermal expansion coefficient) can be specified at up to 10 different temperatures per material type for orthotropic as well as isotropic materials. The program will then linearly interpolate between the values at the given temperatures.

Several numerical techniques have been included in the programs in order to speed up execution time, convergence, and make efficient use of available memory. All of the programs provide for out of core solution of the system of equations so that larger systems can be solved. The memory available for solution is determined by the program and the system of equations is broken up into appropriately sized blocks, stored on file and solved a block at a time using the available core. The shell programs also use a dynamic core allocation scheme which makes maximum use of the available core. This is done by the program surveying the input for the problem to be solved and reserving just the amount of core needed to store the required information and still have a large sized block of core available for solution of the equations. Information will be stored on file as necessary in order to leave a solution block of core available.

The shell programs also have a provision for the user to specify nodes which are fixed (with zero displacement) throughout the problem. The equations associated with these nodes are then eliminated from the system of equations, thus reducing the problem size.

An Aitken acceleration scheme which causes a more rapid convergence to the inelastic solution is built into the 20-noded brick and 8-noded shell programs. This scheme uses the two previous iteration solutions to modify the present iteraction toward the apparent converged solution. This scheme is used with the plasticity and unified constitutive models, but not with the seconday creep model.

These programs cover a large spectrum in sophistication in their ability to predict inelastic response. The problem to be solved, the desired degree of accuracy and available amount of effort will dictate the element type and constitutive model combination to be used. A great deal of overlap occurs, especially for simple problems with simple loadings, but as more complexity is added one combination may be better suited than another. Many capabilities are available in these programs and can be used to great advantage in a wide variety of problems.

2.1 CONSTITUTIVE MODELS

Three separate constitutive models have been implemented in each of the three formulation models. These constitutive models represent a broad perspective in the present state of the art in constitutive modeling, running from a simplified model based on the classical plasticity theory, to a much more sophisticated and advanced treatment of the classical plasticity theory, and finally a unified model where plasticity and creep are viewed together as inelastic action. A separate seconday creep model is combined with the two classical plasticity models in order to expand their capability to the time-dependent domain.

2.1.1 SIMPLE PLASTICITY MODEL

Classical plasticity theories propose the existence of a function describing the onset of yield and leading to an uncoupled rate-independent portion of deformation. There are four common ingredients in the majority of the incremental classical plasticity theories: (1) a yield function distinguishing inelastic and elastic deformation, (2) a relationship between the stress increment tensor and the elastic strain increment tensor, (3) a description of the rate-independent plastic strain increment such as the normality condition, and (4) a work-hardening rule describing the evolution of the yield function under mechanical loading.

The simplified plasticity model is an incremental classical plasticity model which works with effective stresses and strains in the constitutive law and expands these effective quantities to component form through application of the normality condition.

The yield function used in the simplified model is the Von Mises yield function and is given by:

$$\psi(\sigma_{ij}, K) = 3/2\ S_{ij}\ S_{ij} - \sigma_{e_m}^{2} = 0 \qquad (1)$$

where:

$$S_{ij} = \sigma_{ij} - 1/3\ (\sigma_{11} + \sigma_{22} + \sigma_{33}) \qquad (2)$$

is the deviatoric stress tensor, and:

σ_{em} is the maximum effective stress; that is, the yield surface radius.

The above shows that the yield function accounts only for an expansion of the yield surface about the origin (isotropic hardening only). If this function is less than or equal to zero, there is no plastic action. If it is greater than zero, plastic action occurs.

The relationship between the stress state and elastic strain state in the simplified model is characterized by an equation relating effective stress and elastic strain:

$$\varepsilon_e^E = \frac{\sigma_e}{\bar{E}} \tag{3}$$

where:

$$\bar{E} = \frac{3E}{2(1+\nu)} \tag{4}$$

is the effective modulus, which differs from the elastic modulus due to the difference in the definitions of effective stress and effective strain. The effective strain is defined as:

$$\varepsilon_e = \frac{\sqrt{2}}{3}\Big[\left(\varepsilon_{11} - \varepsilon_{22}\right)^2 + \left(\varepsilon_{22} - \varepsilon_{33}\right)^2 + \left(\varepsilon_{33} - \varepsilon_{11}\right)^2 + \frac{3}{2}\left(\varepsilon_{12}{}^2 + \varepsilon_{23}{}^2 + \varepsilon_{31}{}^2\right)\Big]^{1/2} \tag{5}$$

where engineering shear strains are used.

The effective stress is defined as:

$$\sigma_e = \frac{1}{\sqrt{2}}\Big[\left(\sigma_{11} - \sigma_{22}\right)^2 + \left(\sigma_{22} - \sigma_{33}\right)^2 + \left(\sigma_{33} - \sigma_{11}\right)^2 + 6\left(\sigma_{12}{}^2 + \sigma_{23}{}^2 + \sigma_{31}{}^2\right)\Big]^{1/2} \tag{6}$$

The plastic strain increment is calculated using effective quantities as:

$$\Delta\varepsilon_e^P = \Delta\varepsilon_e - \Delta\varepsilon_e^E \tag{7}$$

where:

$\Delta\varepsilon_e$ is the total effective strain increment, and (8)

$$\Delta\varepsilon_e^E = \frac{\sigma_e - \sigma_{em}}{\bar{E}}$$

is the effective strain increment.

To expand this effective plastic strain into component form and also ensure that the plastic strain increment is normal to the yield surface, the Prandtl-Reuss Normality Flow Rule is used. This is given by:

$$d\varepsilon_{ij}^P = d\lambda\ S_{ij} \tag{9}$$

where the scale $d\lambda$ can be calculated from the effective plastic strain increment. To see this, rewrite the Flow Rule as:

$$\left(\frac{2}{3}\right)^2 d\varepsilon_{ij}^P\ d\varepsilon_{ij} = \left(\frac{2}{3}\right)^2 (d\lambda)^2\ S_{ij}\ S_{ij} \tag{10}$$

$$\frac{2}{3}\ d\varepsilon_{ij}^P\ d\varepsilon_{ij}^P = \left(\frac{2}{3}\ d\lambda\right)^2\ \frac{3}{2}\ S_{ij}\ S_{ij} \tag{11}$$

but:

$$\frac{2}{3}\ d\varepsilon_{ij}^P\ d\varepsilon_{ij}^P = \left(d\varepsilon_e^P\right)^2 \tag{12}$$

and:

$$\frac{3}{2}\ S_{ij}\ S_{ij} = (\sigma_e)^2 \tag{13}$$

So:

$$d\varepsilon_e^P = \frac{2}{3}\, d\lambda\, \sigma_e \tag{14}$$

$$d\lambda = \frac{3\, d\varepsilon_e^P}{2\sigma_e} \tag{15}$$

Finally, the work-hardening rule is fully isotropic as mentioned previously. Thus, the maximum effective stress value is retained as the yield surface size for use in the yield function.

For a typical load case, the finite element equations are used to obtain displacements due to the structure loadings and previous inelastic history. These displacements are used to calculate total strain increments. From this, a stress increment is calculated with the initial assumption of no plastic action. The stress state is then used in the yield function to check whether any plastic action occurs in the load step. If not, calculations proceed to the next load case. If plastic action does occur, the effective elastic strain $\Delta\varepsilon_e^E$ is calculated and subtracted from the total effective strain increment to obtain the effective plastic strain increment. This is expanded to component form using the flow rule, and an incremental pseudoforce is then calculated. After going through this procedure over the entire structure, the pseudoforce increments are applied to the structure with the other loads, and the process is repeated until convergence is obtained.

2.1.2 Haisler-Allen Classical Plasticity Model

Professor Walter Haisler and David Allen of Texas A&M developed their classical plasticity theory along the same lines as other classical plasticity theories. However, they have added the capability to accurately handle thermodynamic loadings in conjunction with multiaxial mechanical loadings. Previous classical plasticity models have not been used to model rate-dependent or temperature-dependent media, until recently.

The yield function used in the Haisler-Allen model is assumed to be of the form:

$$F(S_{ij} - \alpha_{ij}) = k^2 \left(\int d\bar{\varepsilon}^P, T \right) \tag{16}$$

where:

S_{ij} = the second Piola-Kirchoff stress tensor

α_{ij} = coordinates of the yield surface center in stress space

k = a characteristic radius dimension describing the size of the yield surface in stress space

$d\bar{\varepsilon}^p$ = the history of equivalent uniaxial plastic strain, and

T = temperature

The yield surface described by this equation defines an area stress space enclosed by the surface where all deformation is elastic and thus recoverable, whereas on this surface, inelastic deformation is allowed. The form of the yield function suggest that the yield surface may both translate kinematically by means of the translation tensor α_{ij} and expand insotropically due to changes in the radius dimension k, thus producing a combined isotropic kineatic work-hardening rule. Note that any thermally induced changes to the yield surface will be totally isotropic in nature.

Differentiation of this general form of the yield function gives a statement of consistency during plastic loading:

$$\frac{\partial F}{\partial S_{ij}} dS_{ij} - \frac{\partial F}{\partial S_{ij}} d\alpha_{ij} - 2k \frac{\partial k}{\partial \bar{\varepsilon}^P} d\bar{\varepsilon}^P - 2k \frac{\partial k}{\partial T} dT = 0 \qquad (17)$$

where the term

$$\frac{\partial F}{\partial S_{ij}}$$

represents

$$\frac{\partial F(S_{ij}-\alpha_{ij})}{\partial S_{ij}}$$

evaluated at $S_{ij}- \alpha_{ij}$, which can be seen to be equivalent to

$$\frac{\partial F}{\partial (S_{ij}-\alpha_{ij})} .$$

Since during neutral loading (one in which the yield surface remains unaltered), the plastic strain increment and $d\alpha_{ij}$ are zero, it is apparent that a statement governing loading can be defined to be:

$$\frac{\partial F}{\partial S_{ij}} dS_{ij} - 2k \frac{\partial k}{\partial T} dT \geq 0 \qquad (18)$$

whereas unloading is described by:

$$\frac{\partial F}{\partial S_{ij}} dS_{ij} - 2k \frac{\partial k}{\partial T} dT < 0 \qquad (19)$$

Two commonly accepted yield functions of the above form are the Tresca and Von Mises yield functions. The Haisler-Allen model uses the Von Mises yield function, which can be described as:

$$F(\sigma_i) = \frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right] = k^2 \qquad (20)$$

where σ_1 σ_2, and σ_3 are principal stresses and k is the current uniaxial yield surface size. This yield condition supposes that yielding is a function of deviatoric stress only. Utilization of this yield function limits the model to analysis of initially isotropic media.

The stress tensor may be related to the elastic strain tensor by:

$$S_{ij} = D^t_{ijmn}\left(E_{mn} - E^P_{mn} - E^C_{mn} - E^T_{mn}\right) \qquad (21)$$

where E_{mn} is the Green-Lagrange strain tensor, D^t_{ijmn} is the temperature-dependent elastic constitutive tensor at time t, and the collected terms in parentheses represent the elastic recoverable strain. In addition, superscripts P, C, and T denote plastic, creep, and thermal strains, respectively.

Since the model is of incremental form, the above equation must be incrementalized. Thus,

$$dS_{ij} = S^{t+\Delta t}_{ij} - S^t_{ij} = D^{t+\Delta t}_{ijmn}\left(E_{mn} - E^P_{mn} - E^C_{mn} - E^T_{mn}\right)^{t+\Delta t} - D^t_{ijmn}\left(E_{mn} - E^P_{mn} - E^C_{mn} - E^T_{mn}\right)^t \qquad (22)$$

which can be rewritten in the form used in the model as:

$$dS_{ij} = D_{ijmn}^{t+\Delta t}\left(dE_{mn} - dE_{mn}^{P} - dE_{mn}^{C} - dE_{mn}^{T}\right) + dD_{ijmn}\left(E_{mn}^{t} - E_{mn}^{Pt} - E_{mn}^{Ct} - E_{mn}^{Tt}\right) \tag{23}$$

where the superscripts t denote parameters measured at the start of a load step and the superscript $t+\Delta t$ denotes measurement at the end of a load step. The terms preceded by d represent the increment in these parameters during the load step.

The flow rule used in the model is the flow rule associated with the Von Mises criterion, and is given by:

$$dE_{ij}^{P} = d\lambda \frac{\partial F}{\partial S_{ij}} \tag{24}$$

where $d\lambda$ is an unknown scalar to be determined by the consistency condition. This equation is commonly called the normality condition. it can be seen that the plastic strain increment is assumed to be a projection on the outer normal to the yield surface, which is scaled by the parameter $d\lambda$.

The final expression required to complete the const_tutive law is the hardening rule. According to Ziegler's modification of the Prager hardening rule, a tensorially correct statement describing translation of the yield surface during a load increment is:

$$d\alpha_{ij} = d_{\mu}\,(S_{ij} - \alpha_{ij}) \tag{25}$$

where the yield surface translation scalar, $d\mu$, contains the temperature and plastic strain history-dependence of the yield surface translation tensor, α_{ij}. The translation scalar is described as:

$$d\mu = \frac{\frac{\partial F}{\partial S_{ij}} dS_{ij} - 2k \frac{\partial k}{\partial T} dT - 2k \frac{\partial k}{\partial \bar{\varepsilon}^P} d\bar{\varepsilon}^P}{(S_{mn} - \alpha_{mn}) \frac{\partial F}{\partial S_{mn}}} \tag{26}$$

These components described above are used to derive the constitutive law used in the Haisler-Allen model. The resulting stress-strain relation is:

$$dS_{ij} = C^t_{ijmn} \left(dE_{mn} - dE^C_{mn} - dE^T_{mn} \right) \tag{27}$$

$$+ dC_{ijmn} \left(E^t_{mn} - E^{Pt}_{mn} - E^{Ct}_{mn} - E^{Tt}_{mn} \right) + dP_{ij}$$

where:

$$C^t_{ijmn} = D^{t+\Delta t}_{ijmn} - \frac{D^{t+\Delta t}_{ijvw} \frac{\partial F}{\partial S_{vw}} \frac{\partial F}{\partial S_{tu}} D^{t+\Delta t}_{tumn}}{\frac{2}{3} H' \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{pq}} + D^{t+\Delta t}_{pqrs} \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{rs}}} \tag{28}$$

$$dC_{ijmn} = dD_{ijmn} - \frac{D^{t+\Delta t}_{ijvw} \frac{\partial F}{\partial S_{vw}} \frac{\partial F}{\partial S_{tu}} dD_{tumn}}{\frac{2}{3} H' \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{pq}} + D^{t+\Delta t}_{pqrs} \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{rs}}} \tag{29}$$

and

$$dP_{ij} = \left[\frac{\sqrt{\frac{2}{3} \frac{\partial F}{\partial S_{tu}} \frac{\partial F}{\partial S_{tu}}} \, D^{t+\Delta t}_{ijmn} \frac{\partial F}{\partial S_{mn}} \frac{\partial \sigma}{\partial T}}{\frac{2}{3} H' \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{pq}} + D^{t+\Delta t}_{pqrs} \frac{\partial F}{\partial S_{pq}} \frac{\partial F}{\partial S_{rs}}} \right] dT \tag{30}$$

Also, H' is the slope of the uniaxial stress-strain diagram for the temperature at the end of the load step, and $\partial\sigma/\partial T$ is the change in uniaxial stress due to a temperature change for the total uniaxial strain at the start of the load step.

It can be seen that the form of the constitutive law is an effective modulus C^t_{ijmn} multiplied by a strain increment added to the change in the effective modulus multiplied by the elastic strain at the start of the step. The term dP_{ij} is a correction term which arises from estimating H' using the uniaxial strain at the start of the step.

The general sequence followed in the implementation of this model in the finite element codes is to solve for displacements from the given loading and previous history. These displacements are then used to calculate an increment in total strain which is fed to the constitutive model along with the temperature increment. A stress increment is then calculated initially assuming zero creep and plastic strain increments. This stress increment is used with the yield function to check for plastic loading. If there is none, the results are correct and the procedure for the load step is complete.

If loading is predicted, the effective modulus (C_{ijmn}), the effective modulus increment (dC_{ijmn}), and the correction factor (dP_{ij}), are calculated and used to determine the stress increment. The yield surface radius (k) is then updated and the translation of the yield surface center ($d\alpha_{ij}$) is calculated to complete the procedure.

This process is carried out over the entire structure resulting in a pseudoforce increment caused by the plastic strain increments. The process is then repeated with the just-calculated pseudoforce increment (along with the other loads applied to the structure) until convergence is achieved.

2.1.3 SIMPLE CREEP MODEL

To make possible the calculation of time-dependent response with the two classical plasticity models, a simplified creep model has been included with these plasticity implementations. This has been done in such a way that the models can be used separately in creep-only or plasticity-only applications, or they may be used simultaneously for certain applications (generally in material testing) where both time-dependent and independent responses may occur. The implementation allows for equal time increments or dynamic time-incrementing as developed for use with the Bodner model.

The simplified creep model is a linear model based on secondary creep only and is represented as:

$$\Delta\varepsilon^c = Q \int_{t_1}^{t_2} (\sigma)^r \, dt \qquad (31)$$

where Q and r are constants. The integration scheme used in the implementation of this model is a trapezoid rule (using the stress value at the beginning and end of the time step) to compute creep strain rates at those times, so that

$$\Delta\varepsilon^c = \left(\frac{\dot{\varepsilon}_1^c + \dot{\varepsilon}_2^c}{2}\right)(t_2 - t_1) \qquad (32)$$

where:

$$\dot{\varepsilon}_1^c = Q(\sigma_1)^r \qquad (33)$$

2.1.3.1 EXAMPLES WITH PLASTICITY/CREEP MODELS

For the classical plasticity models, uniaxial stress cases have been run to check out the constitutive routines. One set of four plasticity-only cases are very useful since they involve both thermal and mechanical loadings in various combinations. Note that the total thermomechanical load is the same for these four cases, but different residual strains result due to the various loading sequences. The following Figures (8 through 13) show the material data used, the four thermomechanical load histories, and the results.

An example of the creep-model-only is shown in which the total strain is imposed and creep strains vary accordingly. This results in a stress relaxation as time increases. The creep coefficients used are chosen to simplify the calculations as Q = 0.44E-8 and r = 1.

Finally, two examples of plasticity and creep combined are shown. The plasticity data used is the same as for the plasticity-only examples, and the creep coefficients are the same as for the creep-only example. The results are encouraging in that the plasticity results are unaffected by the creep, which is the assumption of the classical plasticity theory. The creep results are improved on the total strain, given that the loading is done slowly enough to introduce time-dependent action but quickly enough for time-independent action to occur simultaneously. The use of this option is determined by the user, although some initial examinations of time-step sizing indicate that for slow loadings when creep-only is expected, plasticity does not occur (even with the plasticity calculations included) given small enough time steps.

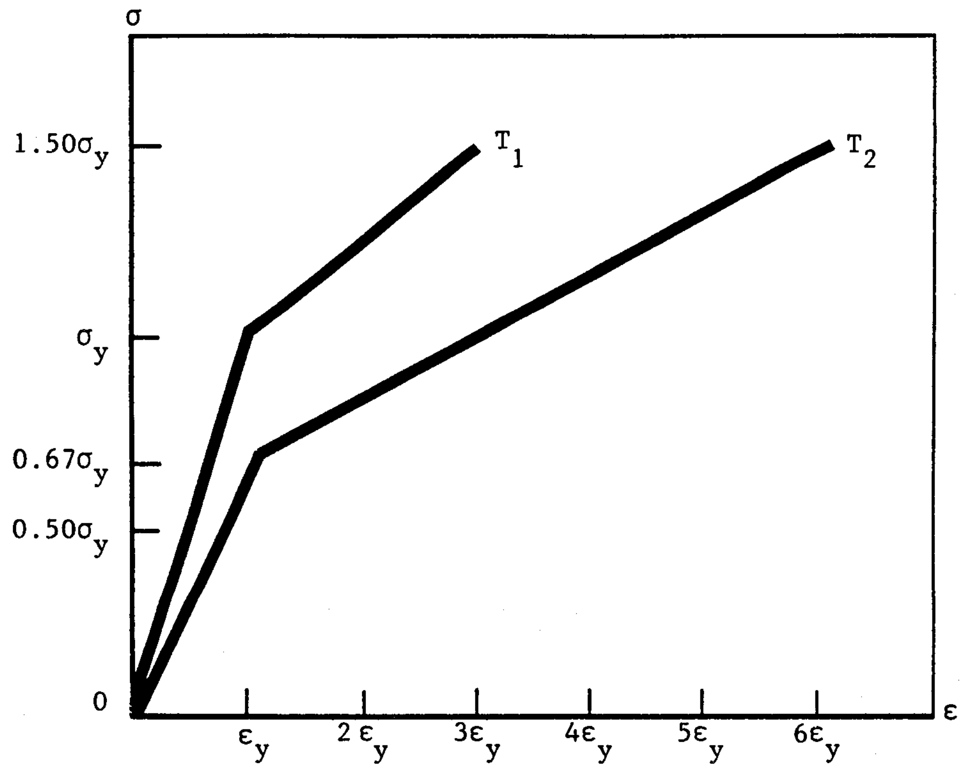


Figure 8. Material Data for Plasticity Example.

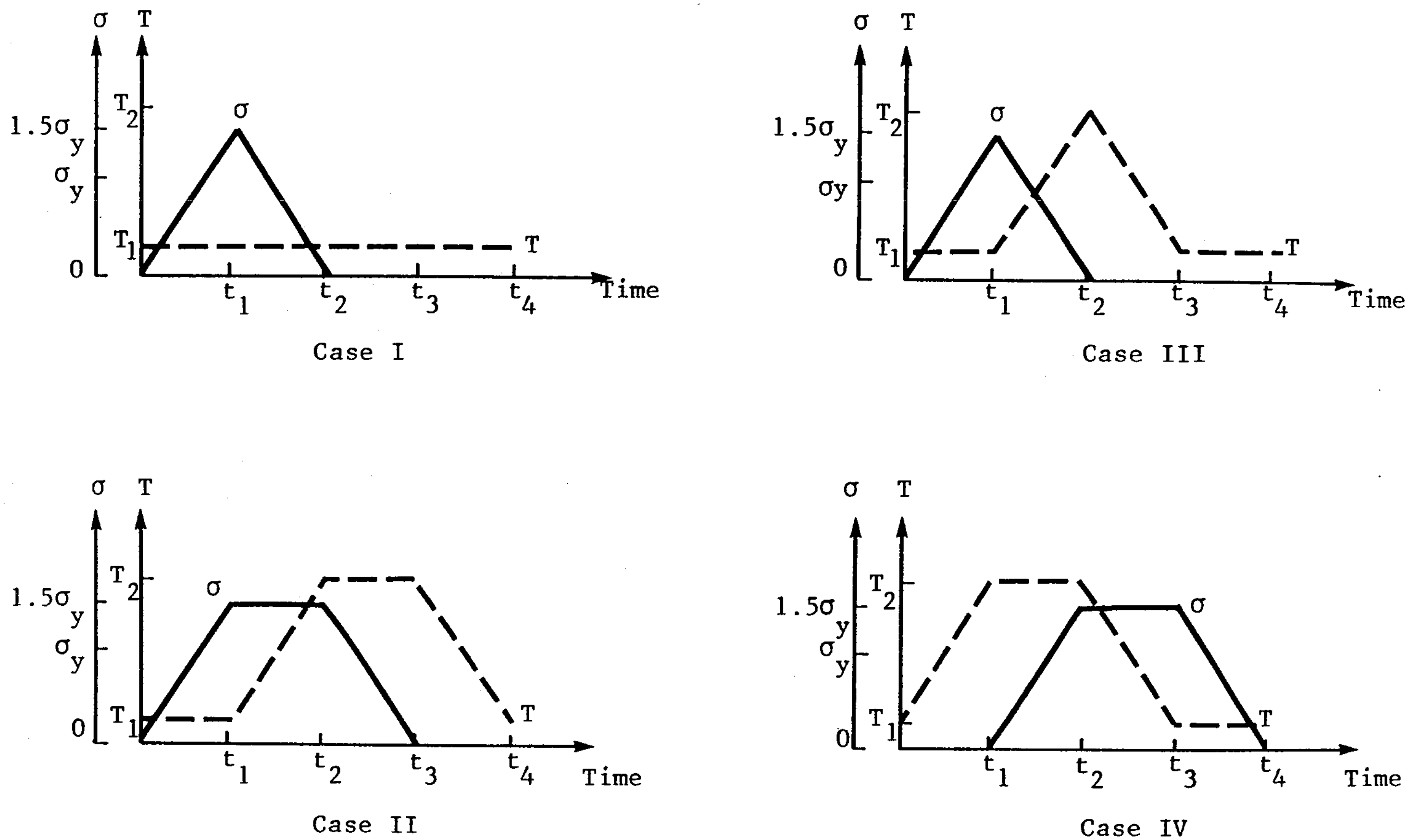


Figure 9. Load Histories for Plasticity Example.

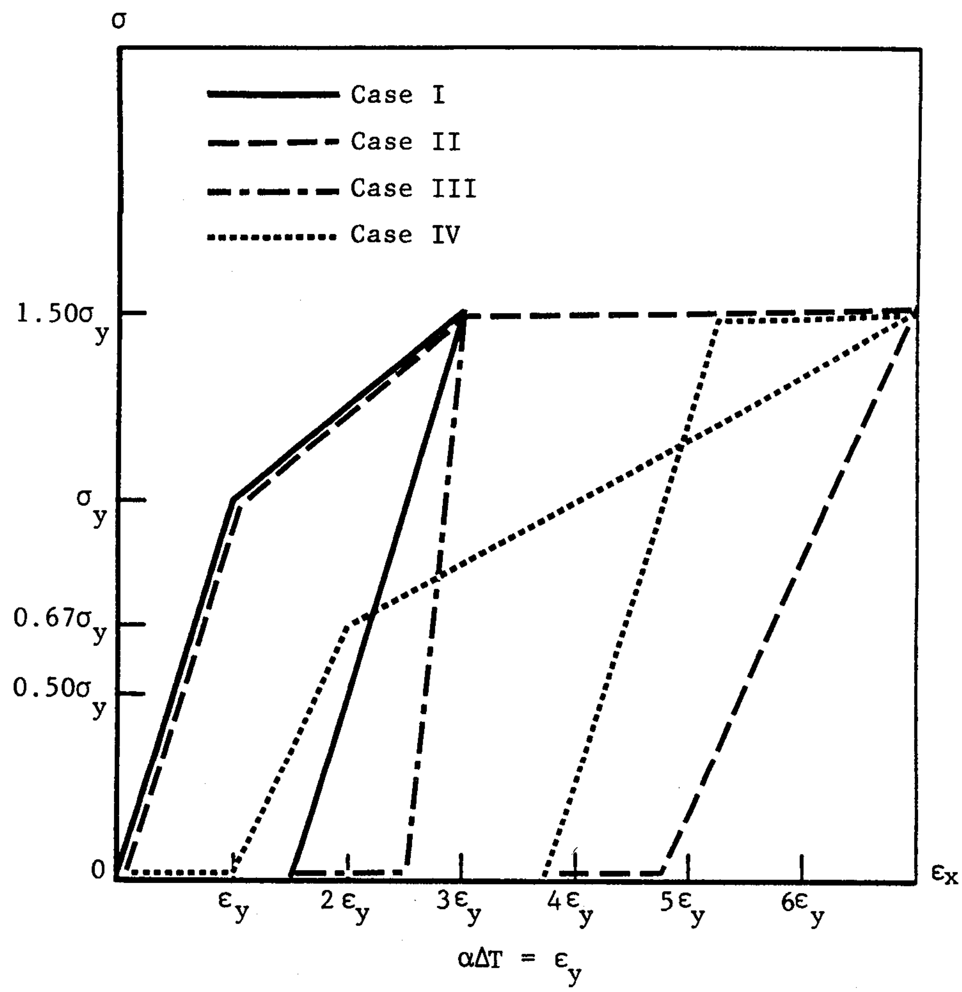


Figure 10. Results of Plasticity Example.

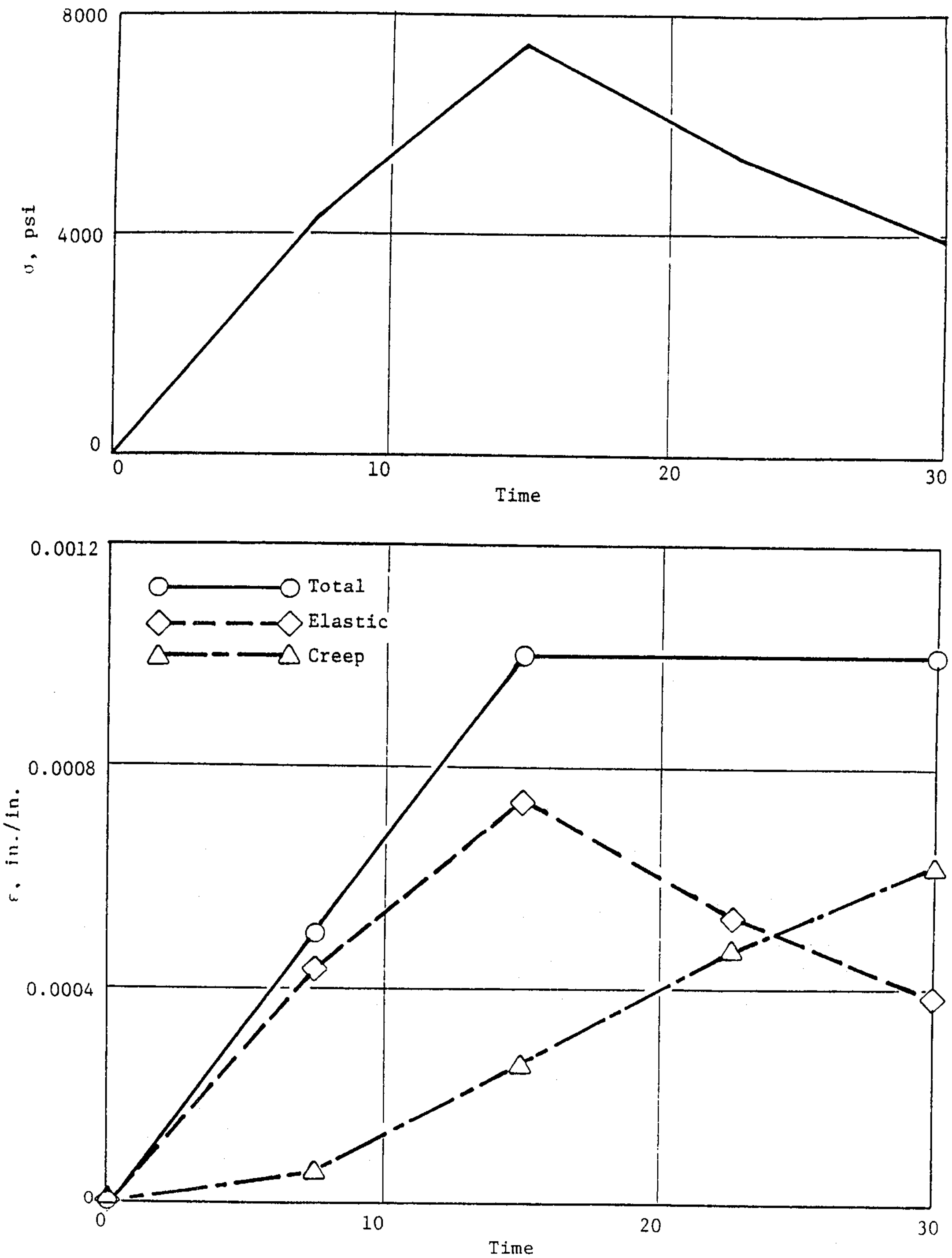


Figure 11. Strain Controlled Creep.

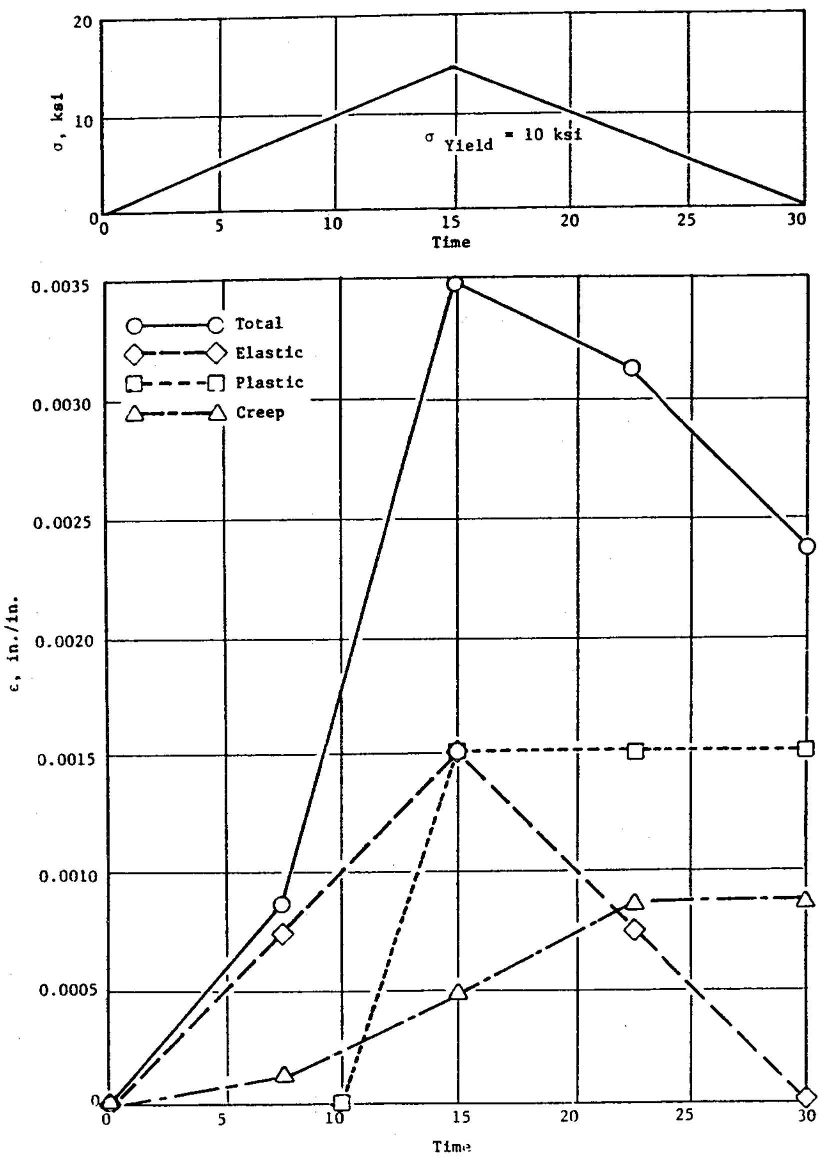


Figure 12. **Stress Controlled Cycling with Combined Plasticity and Creep.**

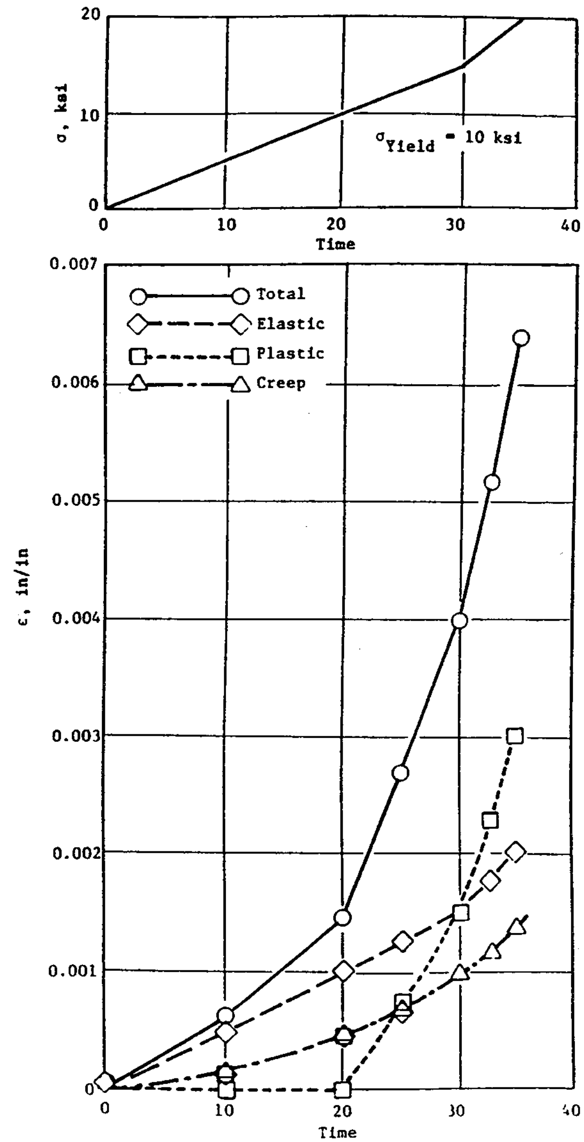


Figure 13. **Stress Controlled Combined Plasticity and Creep.**

2.1.4 BODNER'S UNIFIED MODEL

Bodner's constitutive model is considered a unified model since plastic and creep strains are included in a single inelastic strain measure. Thus, the total strain is the sum of elastic, inelastic, and thermal strains. The inelastic strain rate tensor is defined in a form similar to the Prandtl-Reuss flow rule. It is:

$$\dot{\varepsilon}^{I}_{ij} = Do \exp\left\{-\frac{n+1}{2n}\left[\frac{Z^2}{3J_2}\right]^{n}\right\}\frac{S_{ij}}{\sqrt{J_2}} \tag{34}$$

where Do is the limiting strain rate in shear. The material constant n controls the strain rate sensitivity and J_2 is the second invariant of the deviatoric stress tensor, S_{ij}. The state variable Z is a measure of resistance to inelastic flow and is given an initial value of Zo.

The state variable evolution equation is:

$$\dot{Z} = m\,(Z_1 - Z)\,\dot{W}^{I} - AZ_1\left(\frac{Z - Z_2}{Z_1}\right)^{R} \tag{35}$$

where the first term defines the hardening and the second term characterizes the thermal recovery. The material parameter Z_1 is the maximum value of Z and Z_2 is the minimum value of Z obtained in thermal recovery. The inelastic work rate is $\dot{W}^{I}$ and m, A, and R are material constants.

The Bodner model presented here predicts only isotropic hardening, since Z is a scalar quantity. The material parameters are easily calculated from standard stress strain curves and creep tests, and the numerical implementation is relatively simple.

In order to test the Bodner model, the solution, and dynamic time incrementing schemes discussed in Section 2.3.3 within the context of a finite element code, a two-dimensional model of the benchmark notch specimen (References 10 and 16) containing more than 1,000 constant strain triangles has been run (Figure 14). This code uses Bodner's constitutive model, and the first cycles of Tests 8, 9, and 10 of the benchmark notch program were simulated (Figures 15 through 18). The results were quite satisfying, both in the performance of the constitutive model and in the economy of the solution. On General Electric's Honeywell 6000 computer systems, each of these analyses required approximately 3 hours of CPU time at a cost of about $500 each. With similar codes on the CRAY-1 and the Honeywell 6000, we have experienced speed ratios of about 50 to 1. Therefore, on a machine such as the CRAY-1, we would expect such an analysis to use about five minutes of CPU time. The three analyses required between 205 and 219 time subincrements.

It can be concluded that this code and constitutive model show promise as a design tool.

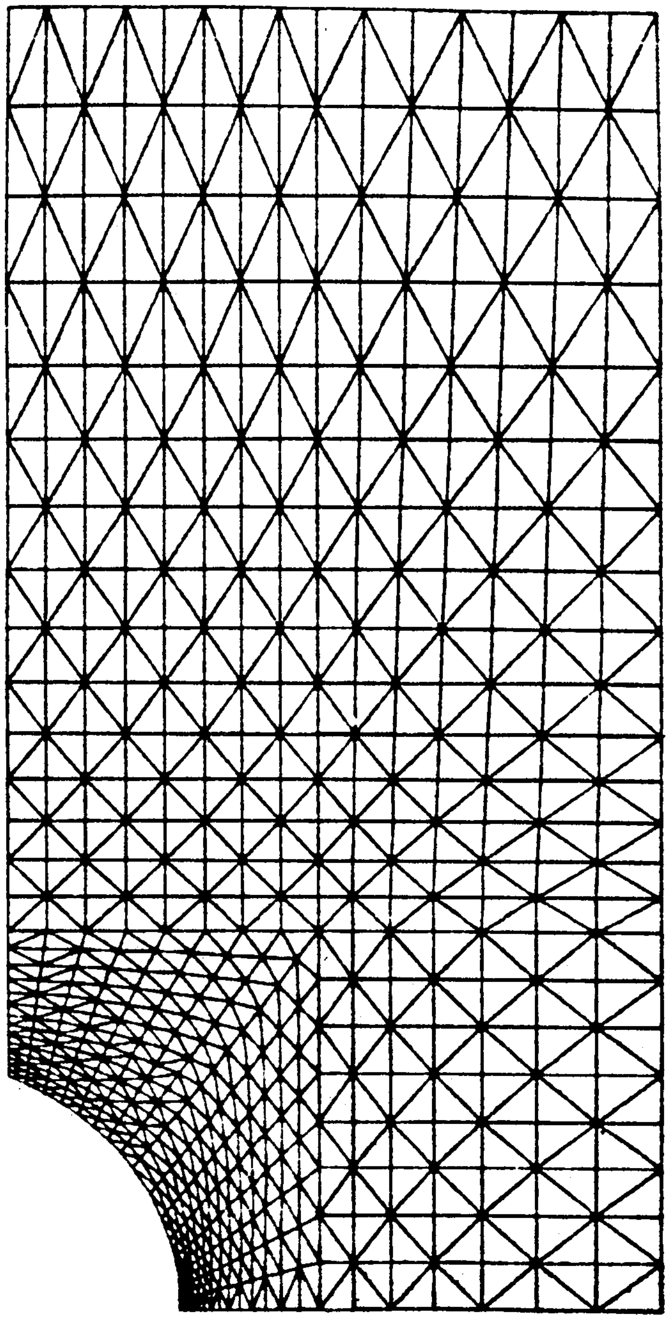


Figure 14. Two-Dimensional Model of Benchmark Notch Specimen.

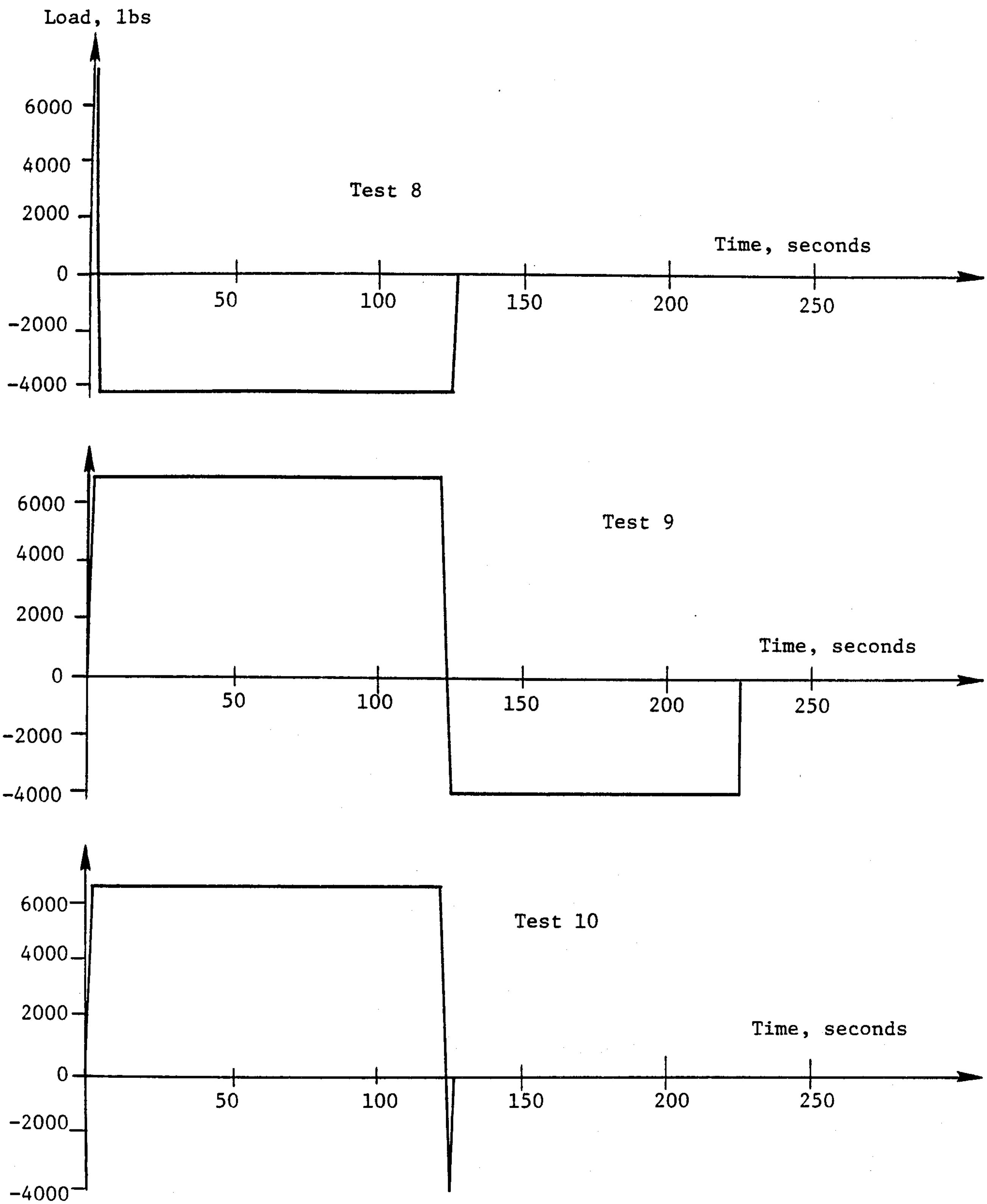


Figure 15. Cycle Tests 8, 9 and 10.

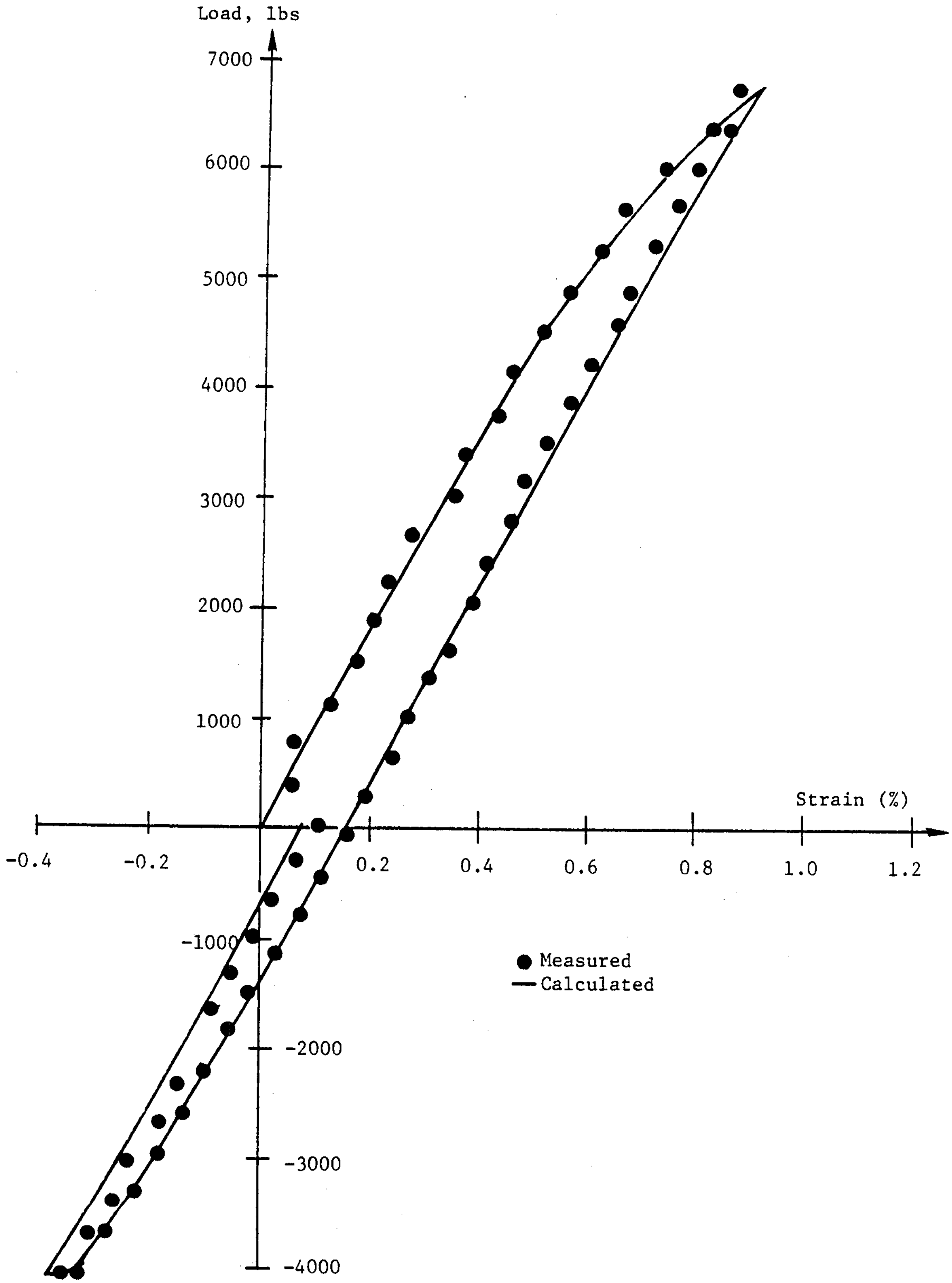


Figure 16. Cycle Test 8.

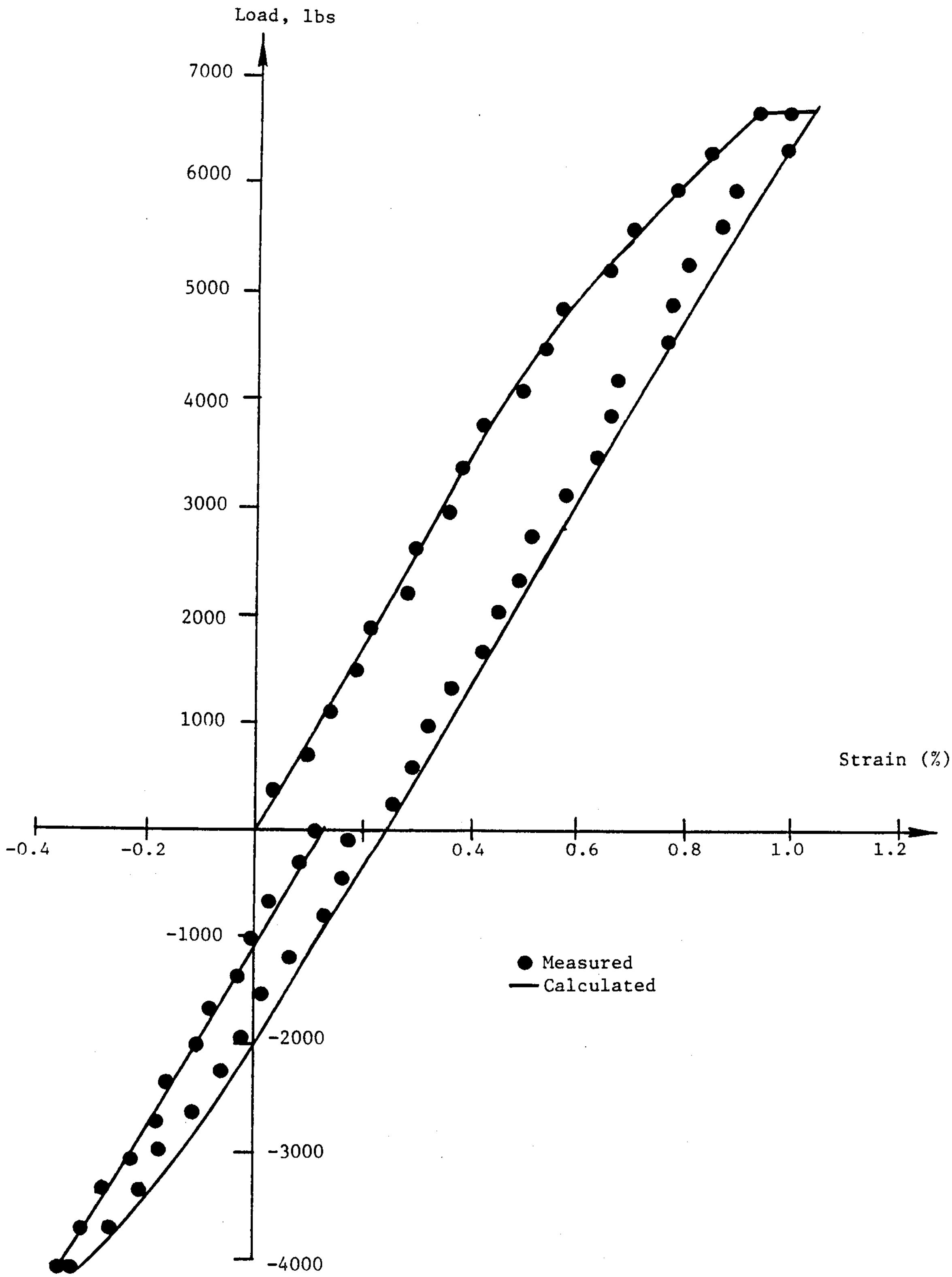


Figure 17. Cycle Test 9.

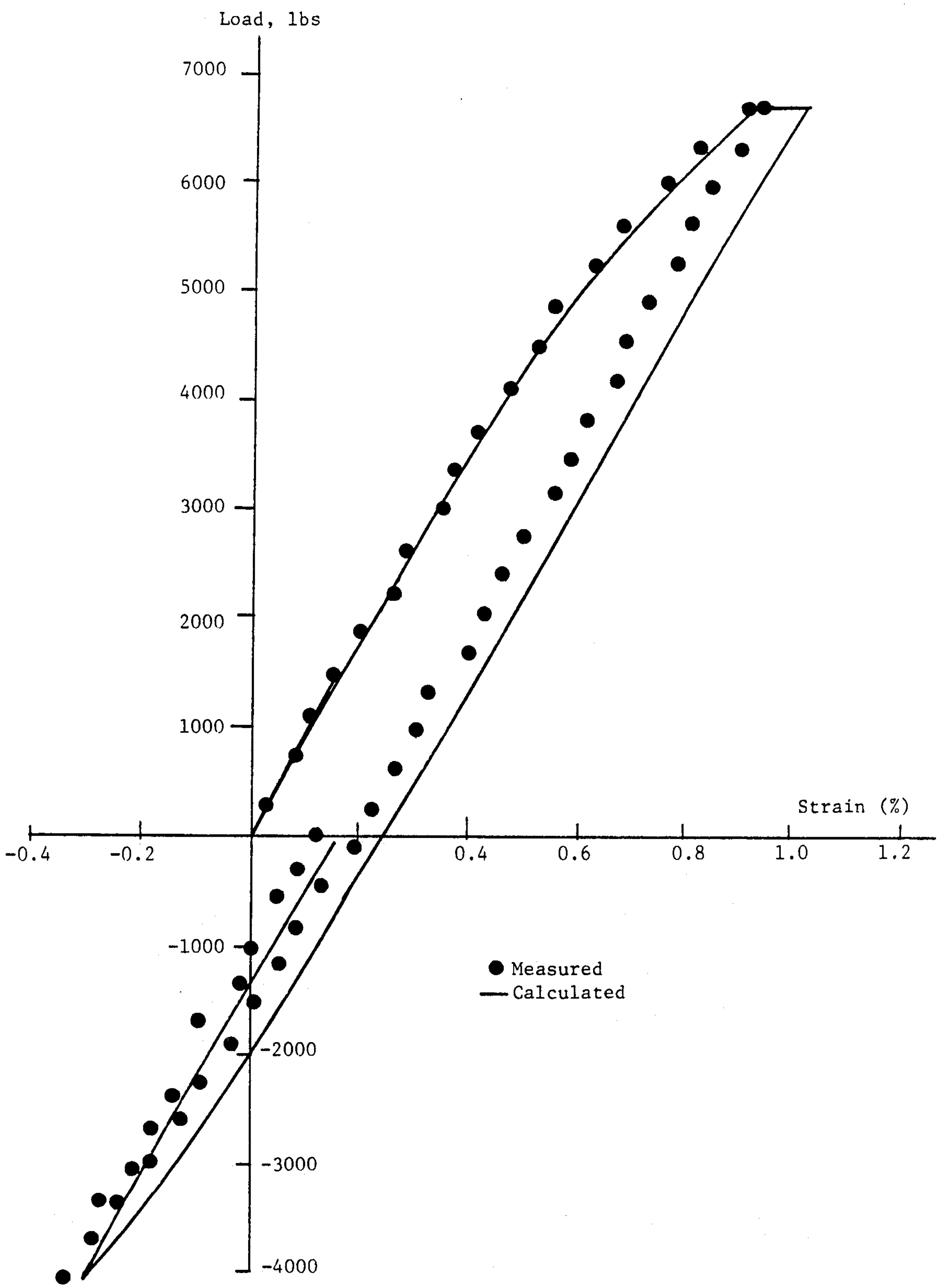


Figure 18. Cycle Test 10.

2.2 FORMULATION MODELS

Three formulation models have been developed, a nine-noded degenerated shell element, an eight-noded degenerated shell element, and a 20-noded isoparametric solid element. The implementation of the theoretical formulation of these elements in computer software has been accomplished in such a way as to optimize their utility for nonlinear analysis. All have been implemented with an out-of-core modified blocked skyline equation solver so that very large, real world problems can be solved. Emphasis has been placed on numerical accuracy, user friendliness, and economy of the situation.

2.2.1 EIGHT-NODE DEGENERATED SHELL

The analysis of shell structures having complex shapes presents an intractable analytical problem in the classical theory of shells. In addition, if the shell structure is thick and shear deformation is significant, the application of classical thin-shell theory becomes questionable. The finite element analysis of such structures is a feasible alternative for obtaining numerical solutions. In modeling shell structures, curved surfaces and faces are often approximated by flat elements with straight sides. For accurate representation of the structure geometry, many of these elements are necessary. The need for elements with curved surfaces is rather obvious.

Isoparametric mapping provides a logical means for developing such elements. The conventional flat-shell elements are often a combination of plane stress and bending elements, but the double-curved shell element has been derived by modifying the three-dimensional, isoparametric, solid-element formulation in a manner consistent with the shell-theory assumptions without restricting behavior to purely thin shells.

The constraint of a straight normal to the midsurface is introduced, and the strain energy corresponding to the direct stress normal to the middle surface is neglected to conform to the shell theory. However, the normal to the middle surface does not remain normal after deformation. This permits the element to experience transverse shear deformations necessary for thick-shell applications.

A temperature-dependent property can be accurately determined using element nodal temperatures. The element loads consist of thermal loading, distributed or uniform pressure loading on element faces, and body forces consisting of centrifugal and gravitational loads. A rotated local Cartesian coordinate system may also be defined at each node of the element. This feature is useful in constraining the rotational degree of freedom about the shell normal.

Consider a 3-D, isoparametric, solid element where linear edges connect higher order faces containing an equal number of nodes. Figure 19 shows a 16-noded, 3-D, isoparametric, solid element where eight-noded parabolic faces are connected by linear edges. The external faces of these elements are curved, but the sections across the thickness are generated by straight lines. For such elements, the pair of Nodes i_t and i_b with given Cartesian coordinates completely describe the element geometry. In 3-D curved shells, this geometry is represented by a middle surface containing the element nodes and the normal to these nodes, such that $\vec{V}_{3i}$ is a unit normal at Node i, and t_i represents the element thickness at Node i (Figure 20 and 21).

Thus, for the 3-D curved-shell-element geometry, we can write:

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_{i=1}^{n} Ni(\xi,\eta) \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix} + \sum_{i=1}^{n} N_i(\xi,\eta) \frac{\xi}{2} t_i V_{3i}$$

where the interpolation shape function for quadratic element is:

Corner nodes N_i $(\varepsilon,\eta) = 1/4(1 + \varepsilon_o)\ (1 + \eta_o)\ (\varepsilon_o + \eta_o - 1)$

Midside Nodes

$\xi_i = 0,\ N_i\ (\xi,\eta) = 1/2\ (1-\xi^2)\ (1 + \eta_o)$

$\eta_i = 0,\ N_i\ (\xi,\eta) = 1/2\ (1+\xi_o)\ (1 - \eta^2)$

$\xi_o \equiv \xi\xi_i,\ \eta_o \equiv \eta\ \eta_i$

DISPLACEMENT FIELD

For the element shown in Figure 20, the pair of nodes i_t and i_b with given translations completely describe element-displacement behavior. For the 3-D curved-shell element, these are reduced into three translations and two rotations at Node i (Figure 22):

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \sum_{i=1}^{n} N_i(\xi,\eta) \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} + \sum_{i=1}^{n} N_i(\xi,\eta) \frac{\xi t i}{2} [\vec{V}_{1i} - \vec{V}_{2i}] \begin{Bmatrix} \alpha_i \\ \beta_i \end{Bmatrix}$$

The unit vectors $\vec{V}_i$, $\vec{V}_{2i}$, and $\vec{V}_{3i}$ define an orthogonal local triad at Node i.

STRESS AND STRAIN CONSTITUTIVE EQUATION

Definition of proper stresses and strains consistent with the shell theory is necessary for deriving element properties. Consider the shell surface defined by ξ = constant. At a point on this surface, construct orthogonal Cartesian axes x', y', and z' such that x'y' lie in the tangent plane and z' is normal to the x'y' plane. The strain components of interest can be defined in the local system x'y'z'.

$$\{\varepsilon'\}^T = [\varepsilon_{x'}, \varepsilon_{y'}, \varepsilon_{x'y'}, \varepsilon_{y'z'}, \varepsilon_{z'x'}]$$

The strain $\varepsilon_{z'}$ has been neglected to satisfy shell assumption. The stresses $\{\sigma'\}$ are related to the strains through the elasticity matrix: $\{\sigma'\} = [c'] \{\varepsilon' - \varepsilon_o'\}$ where the elasticity matrix $[c']$ is a 5x5 symmetric matrix.

NUMERICAL INTEGRATION

The calculation of element matrices and the load vectors require evaluation of the integrals over the element volume or the faces of the element volume. This is accomplished numerically using quadrature. Reduced integration is used in ξ and η directions to improve the bending behavior. A (2x2x2) integration rule has been chosen for the eight-noded, parabolic-shell element.

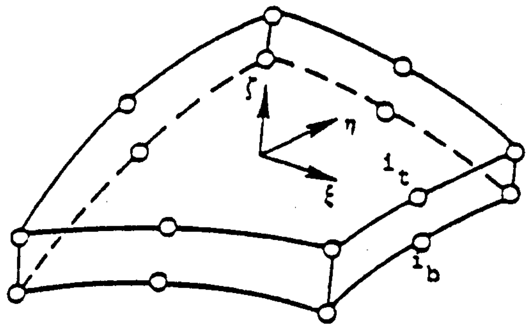


Figure 19. Sixteen-Node Solid Element.

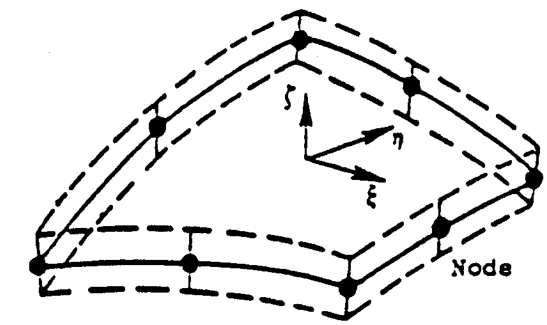


Figure 20. Eight-Node Curved Shell

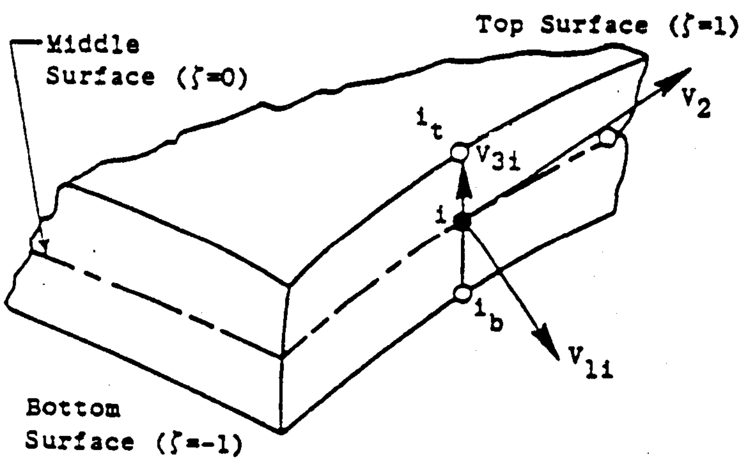


Figure 21. Shell Node i with Normal V_{3i}

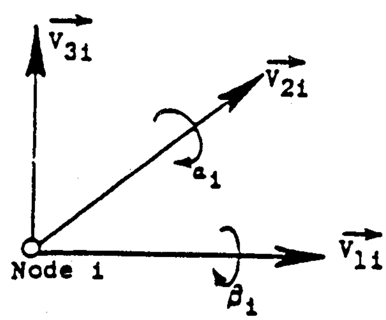


Figure 22. Local Coordinate System and the Rotations α_i, β_i.

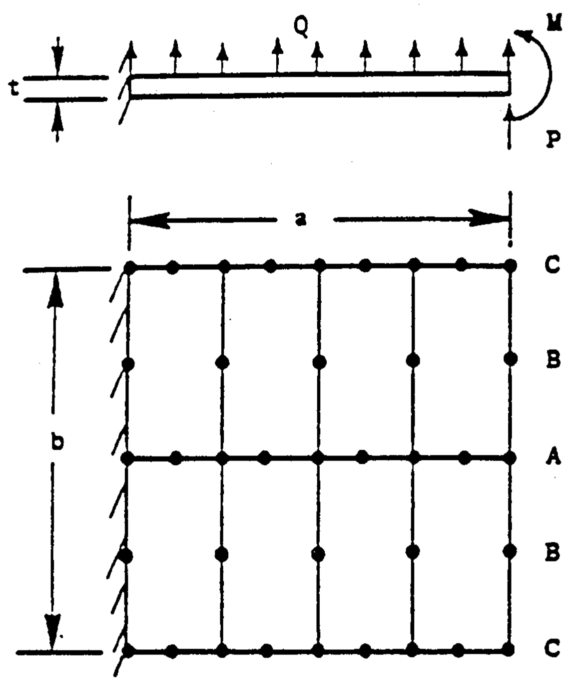
ROTATED LOCAL COORDINATE SYSTEM

With the basic five degrees of freedom (U_i, V_i, W_i, γ_i, β_i), the element does not permit sharp junctions in the structural models. To avoid such difficulty, the rotations γ_i, β_i are transformed into θ_{xi}, θ_{yi}, θ_{zi}. However, the element still does not have rotational stiffness about the normal to the middle surface. Therefore, this rotational degree of freedomn should be constrained at the nodes where the surface is geometrically smooth before solving for the unknown nodal displacements. A local Cartesian coordinate system is established at the nodes requiring such constraining in such a way that one of the axes is normal to the middle surface of the shell. The element stiffness, mass, and the load vectors are transformed into this local coordinate system. The rotational degree of freedom about the axis normal to the middle surface is then deleted before assembly and solution.

EXAMPLE CASES

This element has been implemented in a finite element computer program, and the program debugged, verified, and validated. Three of the test cases are shown in Figures 23, 24, and 25. These cases are very severe tests because the geometries investigated are on the lower end of plate theory and thus involve both beam and plate characteristics for which exact, closed-form solutions do not exist.

To demonstrate its nonlinear capability, the compact tension specimen shown in Figure 26 was modeled (utilizing the line of symmetry) with 30 eight-noded shell elements and 114 nodes. The results of the analysis using the simplified constitutive model is shown in Figure 27. It compared favorably with other nonlinear finite element analyses (CYANIDE, an in-house program) and published data (Reference 7).

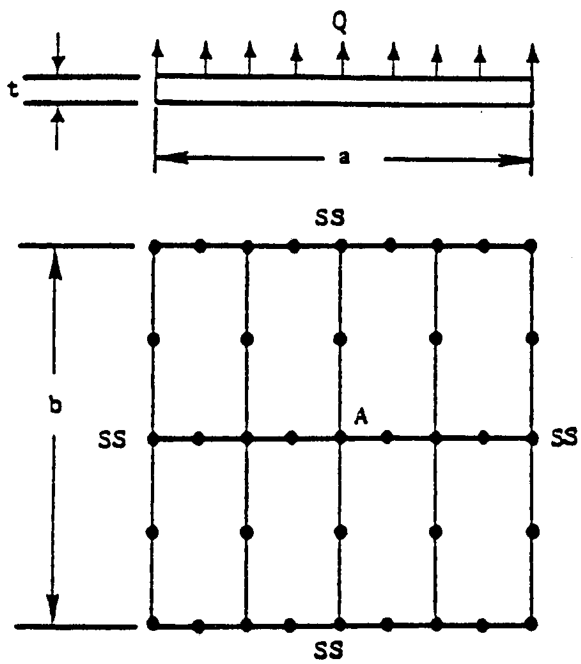


$E = 30 \times 10^6$ psi
$\nu = 0.3$
$a = b = 2$ inch
$t = 0.2$ inch

Figure 23. Example Case 1.

A cantilever plate is subjected to area pressure-load/line-distributed loads as shown above. The square plate was modeled by 8 elements and 37 nodes. The comparison of displacements between beamplate theory and finite-element results are listed below:

Loading Conditions			Vertical Displacements (inch)			
M, (in-lbf/in)	P, (lbf/in)	Q, (psi)	Beam Theory Plate (A, B, and C)	Finite-Element Result A	B	C
1000	0	0	(0.1 to 0.091)	0.0979	0.0963	0.0915
0	0	1000	(0.1 to 0.091)	0.0953	0.0946	0.0938
0	1000	0	(0.1333 to 0.1213)	0.1284	0.1274	0.1252



$E = 30 \times 10^6$ psi

ν 0.3

$a = b = 2$ inch

$t = 0.2$ inch

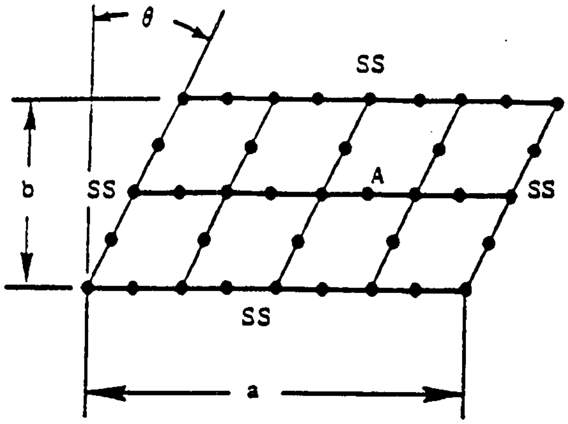
$Q = 1000$ psi

Figure 24. Example Case 2.

A square plate with simply supported (SS) boundary conditions along all edges is subjected to the uniform pressure over the entire plate. The plate was modeled by 8 elements and 37 nodes. The comparison of the maximum vertical displacement at the center of the plate (Location A) was made between the finite-element result and Roark's published solution.

Finite-Element Analysis: $\delta_A = 0.003165$ inch

Roark's Formula: $\delta_A = 0.00296$ inch



$E = 30 \times 10^6$ psi
$\nu = 0.2$
$a = 2$ inch
$b = 1$ inch
$t = 0.2$ inch
$Q = 10^3$ psi

Figure 25. Example Case 3.

A parallelogram plate (skew slab) with all edges simply supported (SS) is under uniform pressure load all over the entire area. The plate was modeled by 8 elements and 37 nodes. At the center of the plate (Location A), the maximum displacement was obtained by finite-element analysis and from Roark's published formula as shown below:

Finite Element Analysis: $\delta_A = 0.00365$ inch

Roark's Formula: $\delta_A = 0.00393$ inch

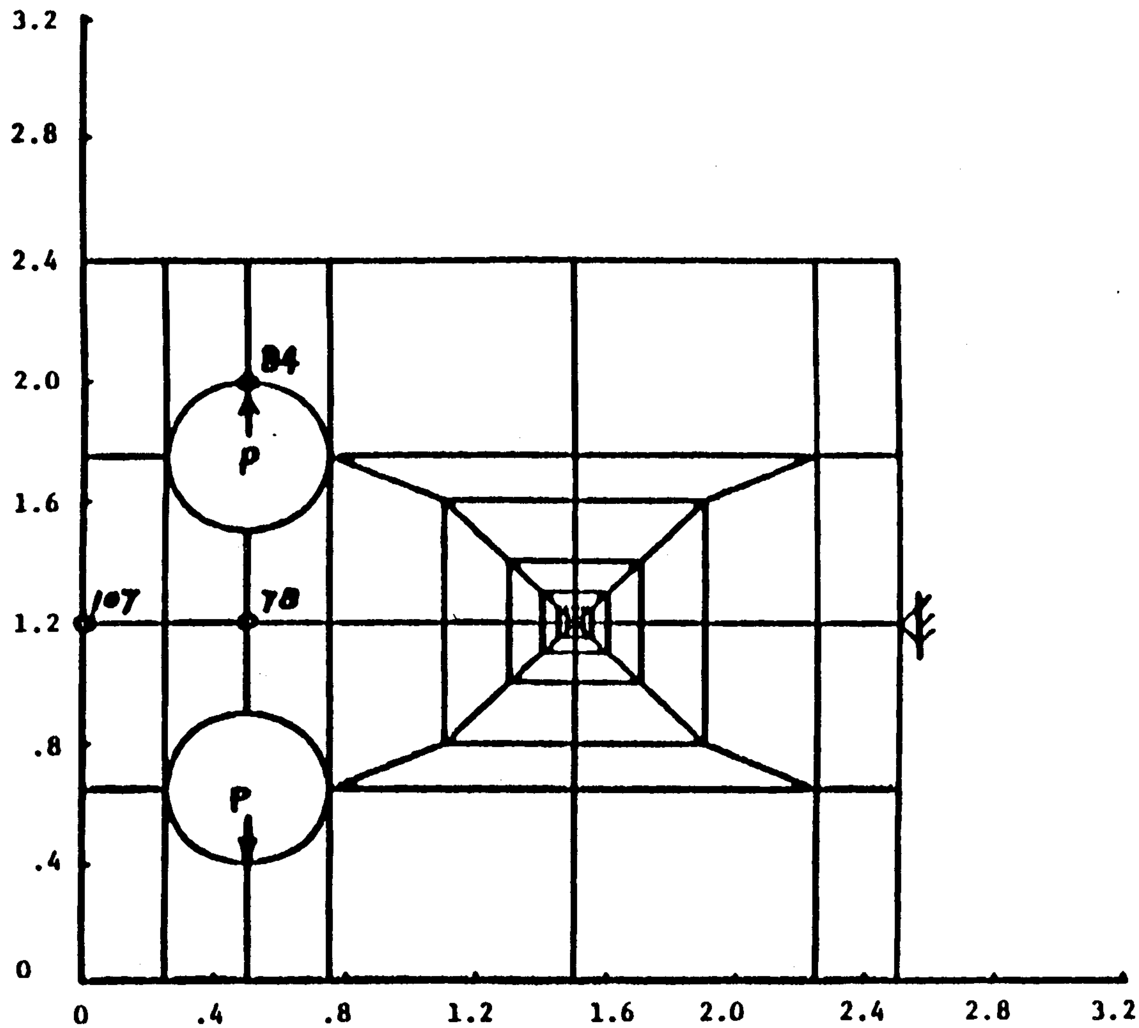
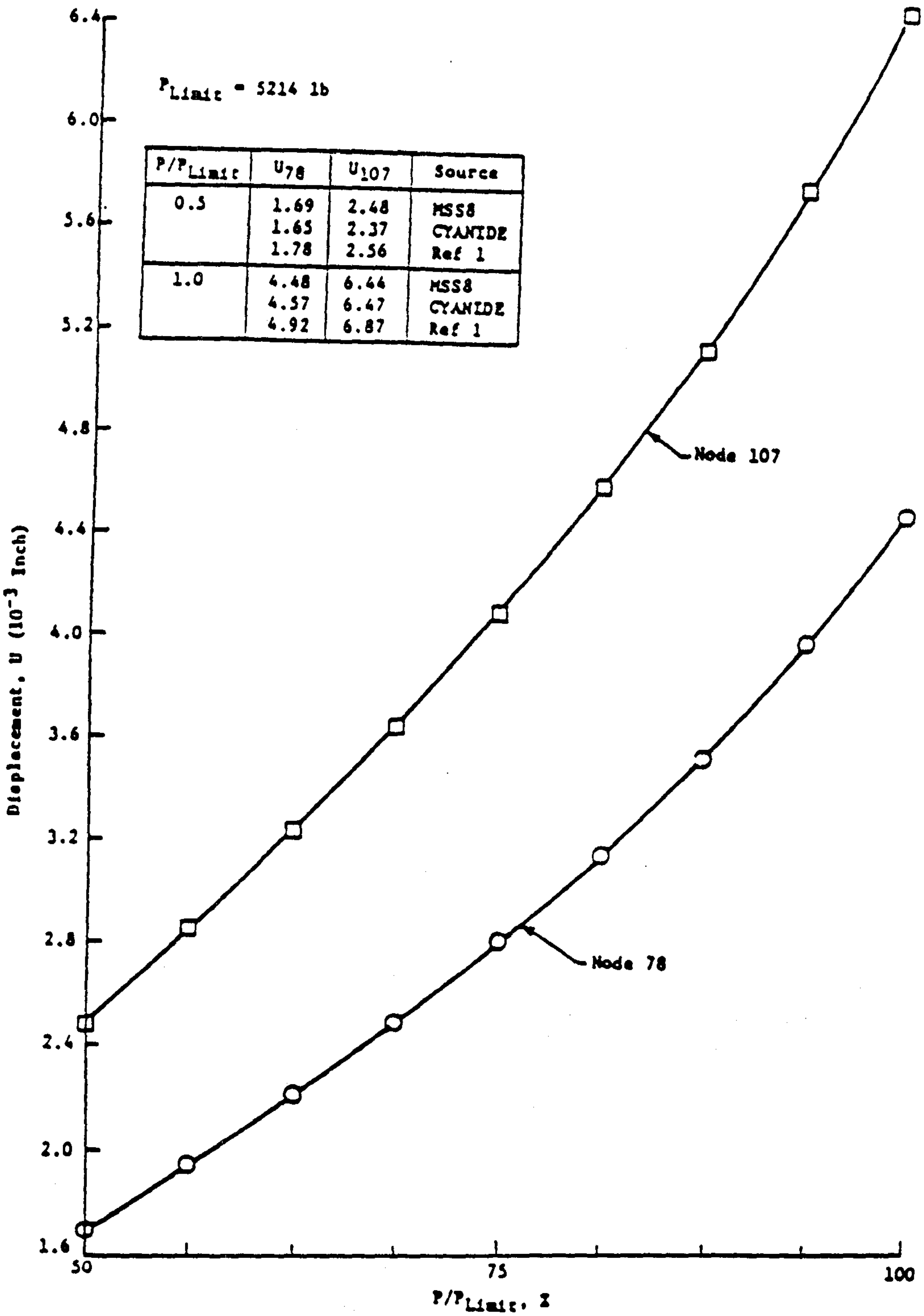


Figure 26. Compact Tension Specimen Model.



P/P_{Limit}	U_{78}	U_{107}	Source
0.5	1.69 1.65 1.78	2.48 2.37 2.56	MSS8 CYANIDE Ref 1
1.0	4.48 4.57 4.92	6.44 6.47 6.87	MSS8 CYANIDE Ref 1

Figure 27. Compact Tension Analysis Results.

2.2.2 NINE-NODE DEGENERATED SHELL

The basis for this element, as it is for most of the others, is the principle of virtual work for displacement increments, written as:

$$\int_V \dot{\vec{\sigma}}\, \delta\dot{\vec{\varepsilon}}\, dv = \int_V \dot{\vec{f}}\, \delta\dot{\vec{u}}\, dv + \int_{s\sigma} \dot{\vec{T}}\, \delta\dot{\vec{u}}\, ds \qquad (36)$$

where $\dot{\vec{u}}$, $\dot{\vec{\varepsilon}}$, $\dot{\vec{\sigma}}$, $\dot{\vec{T}}$ and $\dot{\vec{f}}$ are the displacement-, strain-, stress-, traction- and body force-rate fields, respectively. The displacement-rate field is related to the nodal displacement rates $\{\dot{u}\}$ through shape functions appropriate for each individual element by:

$$\dot{\vec{u}} = [N]\{\dot{u}\} \qquad (37)$$

The shape functions $[N]$ are functions of the spatial coordinates, and are chosen to satisfy admissibility conditions for $\vec{u}$ as well as particular characteristics for each element. The strain field is found by taking the appropriate combination of partial derivatives of $\vec{u}$ with respect to the spatial coordinates, and may be written as:

$$\dot{\vec{\varepsilon}} = [N']\{\dot{u}\} \qquad (38)$$

The stress-strain law is in the form (see Section 2.1)

$$\dot{\vec{\sigma}} = [E]\dot{\vec{\varepsilon}} - \dot{\vec{\tau}} \qquad (39)$$

Substituting (37)-(39) into (36) and taking variations with respect to $\{\dot{u}\}$ yields the element equations:

$$[K]\ \{\dot{u}\} = \{\dot{F}^{IE}\} + \{\dot{F}^{B}\} + \{\dot{F}^{T}\} + \{\dot{F}\} \tag{40}$$

where:

$$[K] = \int_v [N']^T\ [E]\ [N']dV \tag{41.a}$$

$$\{\dot{F}^{IE}\} = \int_v [N']^T\ \dot{\vec{\tau}}\ dV \tag{41.b}$$

$$\{\dot{F}^{B}\} = \int_v [N]^T\ \dot{\vec{f}}\ dV \tag{41.c}$$

$$\{\dot{F}^{T}\} = \int_{s\sigma} [N]^T\ \dot{\vec{T}}\ dS \tag{41.d}$$

and where $\{F\}$ are the nodal force rates exerted on the element by its neighbors. Generally, some of the body force field is due to structural vibration. This ultimately yields a mass matrix given by

$$[M] = \int_v [N]^T\rho\ [N]\ dV \tag{41.e}$$

where ρ is the mass density of the element. The objective of any of the element packages is to determine five expressions (41) appropriate for a given element, and from geometric, material, and load data, determine numerical values for the entries in the particular vectors and matrices.

The nine-node degenerated shell element formulation is a modification of the element developed by Chang, et al. (References 3 and 4). The Figures illustrate the geometry of the element. The middle surface of the shell is generated by mapping a 2x2 square into the surface using:

$$\dot{\vec{r}}_o = (x_o, y_o, z_o) = [N(\xi, \eta)] \{\{x\}, \{y\}, \{z\}\} \qquad (42)$$

where (x_o, y_o, z_o) represents a point on the midsurface of the shell and $\{\{x\}, \{y\}, \{z\}\}$ is the collection of node point coordinates, one of the given bits of information in the element development. The shape functions $[N]$ are the bequadratic shape functions for the 2x2 square. Also given at each node point are the shell thicknesses $\{h\}$. The thickness is assumed to be measured locally normal to the shell midsurface. We determine the normal direction by first obtaining e^o_ξ and e^o_η, vectors tangent to the midsurface at a particular point, and taking their cross product. These vectors are found by:

$$\vec{e}^{\,o}_\xi = \frac{\partial \vec{r}o}{\partial \xi} = \left[\frac{\partial N}{\partial \xi}\right] \{\{x\}, \{y\}, \{z\}\} \qquad (43)$$

$$\vec{e}^{\,o}_\eta = \frac{\partial \vec{r}o}{\partial \eta} = \left[\frac{\partial N}{\partial \eta}\right] \{\{x\}, \{y\}, \{z\}\}$$

The normalized cross-product is labeled v_3, defined in symbols as:

$$\vec{V}_3 = (\vec{e}^{\,o}_{\xi} \times \vec{e}^{\,o}_{\eta})/|e_{\xi} \times e_{\eta}| = (v_{3,1}, v_{3,2}, V_{3,3}) \quad (44)$$

The nondimensionalized coordinate in this direction is labeled ξ. Thus, any point in the shell is represented as:

$$\vec{r} = (x,y,z) = [N(\xi,\eta)]\{\{x + \tfrac{1}{2}\zeta hv_{3,1}\}, \{y + \tfrac{1}{2}\zeta hv_{3,2}\}, \{z + \tfrac{1}{2}\zeta hv_{3,3}\}\} \quad (45)$$

An individual element of the vector $\{x = 1/2\ 3hv_{3,1}\}$, for instance, is $x_i + 1/2\ 3h_i\ v^i_{3,1}$, where x_i, h_i, and v^i_3 are the x, coordinate, thickness, and x-component of the v_3 vector at node i, respectively. The z dependence, then, is assumed to be interpolated in the same way as the midsurface coordinates, without using the (ξ,η) explicit function implied by (44). The three directional vectors for the mapping (45) are:

$$\vec{e}\xi = \frac{\partial \vec{r}}{\partial \xi} = \vec{e}^{\,o}_{\xi} + \frac{1}{2}\zeta\left[\frac{\partial N}{\partial \xi}\right]\{\{hv_{3,1}\}, \{hv_{3,2}\}, \{hv_{3,3}\}\} \quad (46)$$

$$\vec{e}\eta = \frac{\partial \vec{r}}{\partial \eta} = \vec{e}^{\,o}_{\eta} + \frac{1}{2}\zeta\left[\frac{\partial N}{\partial N}\right]\{\{hv_{3,1}\}, \{hv_{3,2}\}, \{hv_{3,3}\}\}$$

$$\vec{e}\zeta = \frac{\partial \vec{r}}{\partial \zeta} = \frac{1}{2}\ [N]\ \{\{hv_{3,1}\}, \{hv_{3,2}\}, \{hv_{3,3}\}\}$$

The Jacobian matrix (J) is formed by:

$$[J] = \begin{bmatrix} \vec{e}\xi \\ \vec{e}\eta \\ \vec{e}\zeta \end{bmatrix} \qquad (47)$$

In general, (J) is a function of (ξ, η, ζ): however, interest in (J) will be at the nodal points i (i=1,...,9), where z = 0, and above and below the nodal points, where ζ = 1, for a total of 27 points. These 27 points will be the numerical integration points for the element.

The (ξ, η, ζ) coordinates are convenient to describe the geometry. In addition, $\vec{e}_z$ is parallel to $\vec{V}_3$ at the nodal points. Note that the e_z and e_η are not necessarily perpendicular to each other, (and away from the midsurface, probably not perpendicular to eζ), and thus do not form a local Cartesian system at any point. However, from a shell analysis point of view, it is desirable to distinguish between in-plane behavior. In order to accomplish this, the (1, 2, 3) coordinate system is introduced where $\vec{v}_3$ is the unit vector orthogonal to the midsurface, and $\vec{v}_1$ and $\vec{v}_2$ are defined orthogonal to each other in the plane of the midsurface. The v_1 vector is defined by normalizing the cross product of $\vec{v}_3$ and the Cartesian unit vector most perpendicular to it; then $\vec{v}_2 = \vec{v}_3 \times \vec{v}_1$.

At each node point of the element, there is defined the six degrees of freedom (DOF) for that node point. They are the three translations U_x, U_y, and U_z and the three rotations θ_x, θ_y, and θ_z. Note that these are defined relative to global DOF to local DOF in the (1, 2,

3) system. Actually, the local translations will be identical to the global ones, and only the rotations will be transformed. As in most shell analysis, the DOF θ_3 will be suppressed. The transformation at each node point is accomplished through the matrix T_i given as:

$$\overleftrightarrow{[T_i]} = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & v^i_{1,1} & v^i_{1,2} & v^i_{1,3} \\ 0 & 0 & 0 & v^i_{2,1} & v^i_{2,2} & v^i_{2,3} \\ 0 & 0 & 0 & v^i_{3,1} & v^i_{3,2} & v^i_{3,3} \end{bmatrix} \tag{48}$$

The collection of $[T_i]$'s along a diagonal is denoted as (T); thus, for the element

$$\left\{ \dot{u}^{local} \right\} = [T] \left\{ u^{global} \right\} \tag{49}$$

The [T] matrix is 54x54 and there are 54 DOF's for this element.

Another useful matrix is the tensor transformation matrix $\overleftrightarrow{[Ti]}$ for node point i, given as:

$$
\overleftrightarrow{[Ti]} = \begin{bmatrix}
(v^i_{1,1})^2 & (v^i_{1,2})^2 & (v^i_{1,3})^2 & v^i_{1,1}\,v^i_{1,2} & v^i_{1,2}\,v^i_{1,3} & v^i_{1,3}\,v^i_{1,1} \\
(v^i_{2,1})^2 & (v^i_{2,2})^2 & (v^i_{2,3})^2 & v^i_{2,1}\,v^i_{2,2} & v^i_{2,2}\,v^i_{2,3} & v^i_{2,3}\,v^i_{2,1} \\
(v^i_{3,1})^2 & (v^i_{3,2})^2 & (v^i_{3,3})^2 & v^i_{3,1}\,v^i_{3,2} & v^i_{3,2}\,v^i_{3,3} & v^i_{3,3}\,v^i_{1,1} \\
2v^i_{1,1}\,v^i_{2;1} & 2v^i_{1,2}\,v^i_{2,2} & 2v^i_{1.3}\,v^i_{2,3} & v^i_{1,1}\,v^i_{2,2} + v^i_{2,1}\,v^i_{1,2} & v^i_{1,2}\,v^i_{2,3} + v^i_{2,2}\,v^i_{1,3} & v^i_{1,3}\,v^i_{2,1} + v^i_{2,3}\,v^i_{1,1} \\
2v^i_{2,1}\,v^i_{3,1} & 2v^i_{2,2}\,v^i_{3,2} & 2v^i_{2,3}\,v^i_{3,3} & v^i_{2,1}\,v^i_{3,2} + v^i_{3,1}\,v^i_{2,2} & v^i_{2,2}\,v^i_{3,3} + v^i_{3,2}\,v^i_{2,3} & v^i_{1,3}\,v^i_{2,1} + v^i_{2,3}\,v^i_{1,1} \\
2v^i_{3,1}\,v^i_{1,1} & 2v^i_{3,2}\,v^i_{1,2} & 2v^i_{3,3}\,v^i_{1,3} & v^i_{3,1}\,v^i_{1,2} + v^i_{1,1}\,v^i_{3,2} & v^i_{3,2}\,v^i_{1,3} + v^i_{1,2}\,v^i_{3,3} & v^i_{3,3}\,v^i_{1,1} + v^i_{1,3}\,v^i_{3,1}
\end{bmatrix} \tag{50}
$$

The degrees of freedom are defined at the midsurface of the shell. To find the displacement field at any point in the shell, use:

$$\vec{\dot{u}} = [N(\xi, \eta)]\,[Q]\{\dot{u}^{\,local}\} \tag{51}$$

where here, $[N]$ is the biquadratic repeated over three rows, thus making $[N]$ a 3x27 matrix, and $[Q]$ is 27x54 made up of nine 3x6 submatrices $[Q_i]$ arranged along its diagonal, each $[Q_i]$ being:

$$[Q_i] = \begin{bmatrix} 1 & 0 & 0 & -\frac{1}{2}\zeta h_i\, v^i_{2,1} & \frac{1}{2}\zeta h_i\, v^i_{1,1} & 0 \\ 0 & 1 & 0 & -\frac{1}{2}\zeta h_i\, v^i_{2,2} & \frac{1}{2}\zeta h_i\, v^i_{1,2} & 0 \\ 0 & 0 & 1 & -\frac{1}{2}\zeta h_i\, v^i_{2,3} & \frac{1}{2}\zeta h_i\, v^i_{1,3} & 0 \end{bmatrix} \tag{52}$$

Note that $\vec{u}$ is defined in terms of the global directions and the zero sixth column of $[Q_i]$ indicates the independence of $\vec{u}$ on θ_3. Furthermore, $[Q_i]$ clearly represents the usual shell assumption of linear variation of the displacement field through the thickness.

Strains are calculated in Cartesian coordinates first. The relation between the differential operators in (x, y, z) and (ξ, η, ζ) are:

$$[f'] = \begin{bmatrix} \partial/\partial x \\ \partial/\partial y \\ \partial/\partial \zeta \end{bmatrix} = [J]^{-1} \begin{bmatrix} \partial/\partial \xi \\ \partial/\partial \eta \\ \partial/\partial \zeta \end{bmatrix} \tag{53}$$

Denoting as $[F']$ a 9x3 matrix with 3 $[f']$ arranged on the "main diagonal," and $[D]$, a 6x9 Boolean matrix denoting a linear combination of partial derivatives of $\vec{u}$, the strain displacement relation is written as:

$$\dot{\vec{\varepsilon}} = [d] \; ([F'] \; [N] \; [Q]) \; [T] \left\{ \dot{u}^{\,global} \right\} \tag{54}$$

where use has been made of (49). The combination of matrices on the right-hand side of (54) is denoted as $[\beta^{global}]$.

The strain relative to (x, y, z) is generally not interesting from a physical point of view. It is desirable to obtain the strain in the (1, 2, 3) system where it physically has greater meaning. Thus, using (50),

$$[\beta^{\,local}_{ij}] = [\overleftrightarrow{T}_i] \; [\beta^{\,global}_{ij}] \tag{55}$$

where the i subscript refers to the i^{th} nodal point, and the j subscript refers to the j^{th} surface (middle, upper, or lower).

The constitutive law (39) must be modified to take into account the mechanics of materials assumption that $\sigma_{33} = 0$. With [E] and supplied in the (1, 2, 3) system for the 27 integration points, a partitition of (39) is made as follows:

$$\begin{Bmatrix} \dot{\sigma}_c \\ 0 \end{Bmatrix} = \begin{bmatrix} E_{c3} & E_{cc} \\ E_{3c} & E_{33} \end{bmatrix} \begin{Bmatrix} \dot{\epsilon}_c \\ \dot{\epsilon}_{33} \end{Bmatrix} - \begin{Bmatrix} t_c \\ t_{33} \end{Bmatrix} \qquad (56)$$

where σ_c are the five non-zero stress-value fields, $\dot{\epsilon}_c$ the corresponding five strain fields, and the dimensions of E_{cc}, E_{c3}, E_{3c}, E_{33}, t_c, and $\dot{t}_{33}$, are 5x5, 5x1, 1x5, 5x1, 1x1, 5x1, and 1x1, respectively. Solving the lower partition of ϵ_{33} and using it in the upper partition yields the modified relation:

$$\left\{\dot{\sigma}_c\right\} = \left[\hat{E}_{cc}\right]\left\{\dot{\epsilon}_c\right\} - \dot{\hat{\tau}}_c \qquad (57)$$

where:

$$\left[\hat{E}_{cc}\right] = \left[E_{cc}\right] - \left[E_{c3}\right]\left[E_{33}\right]^{-1}\left[E_{3c}\right] \qquad (58)$$

$$\dot{\hat{\tau}}_c = \dot{\tau}_c - \left[E_{c3}\right]\left[E_{33}\right]^{-1}\dot{\tau}_{33} \qquad (58b)$$

Furthermore, it is convenient to define $[\partial_{ijc}^{local}]$ as $[\partial_{ij}^{local}]$, with the row corresponding to ε_{33} deleted; then:

$$[\partial_{ijc}^{local}] = [\hat{E}_{cc}]\,[\zeta_{ijc}^{local}] \tag{59}$$

The matrix $[\partial_{ij}^{local}]$ can be defined as $[\partial_{ijc}^{local}]$ augmented by a row of zeros corresponding to $\dot{\tau}_{33} = 0$. Finally, the stress in global coordinates is expressed as:

$$\dot{\sigma}_{ij}^{global} = \overleftrightarrow{[T]}^{-1}\left(\left[\partial_{ij}^{local}\right]\left\{\dot{u}^{global}\right\} - \dot{\tau}_{ij}^{local}\right) \tag{60}$$

Element matrix and vector calculations are now ready to begin. For instance for the stiffness matrix, the integrand $[k_{ij}]$ at the $(i,j)^{th}$ integration point is:

$$[k_{ij}] = [\zeta_{ijc}^{local}]\;[\partial_{ijc}^{local}] \tag{61}$$

The volume integration will be taken numerically by approximating the integrand in (41.a) by evaluation of the integrand at the 27 integration points and then interpolating these using the triquadratic shape functions in (ξ, η, ζ). The shape functions are integrated exactly over the nondimensionalized cube:

$$\text{If}\quad a_{ij} \equiv \int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1}[N_{ij}(\xi, \eta, \zeta)]\,d\zeta\,d\eta\,d\xi \tag{62}$$

where $[N_{ij}]$ is the appropriate shape function for the $(i,j)^{th}$ integration point, then:

$$[K] = \Sigma\, \alpha_{ij}\, [K_{ij}]\, \det\, [J_{ij}] \tag{63}$$

The vector $\{\dot{F}^{IE}\}$ is calculated in a similar manner; defining

$$\{\dot{F}_{ij}\} = [\mathfrak{z}_{ijc}^{\,local}]^T\, \dot{\tau}_c^{\,local} \tag{64}$$

then:

$$\{\dot{F}^{IE}\} = \Sigma_{i,j}\, \alpha_{ij}\, \{\dot{F}_{ij}\}\, \det\, [J_{ij}] \tag{65}$$

The mass matrix calculated this way yields what appears to be a lumped-typed mass matrix.

In general, the value of the body force field is either given or can be calculated at every integration point. Let $\{f_b^{ij}\}$ be the body force load at the $(i,j)^{th}$ integration point. If $\{\dot{F}_B^i\}$ is defined as 6x1 subvector corresponding to the i^{th} node's contribution $\{\dot{F}_B\}$,

$$[\dot{F}_B^i] = \sum_J \alpha_{ij}\, [T_i]^T\, [Q_{ij}]^T\, \{\dot{f}_B^{ij}\}\, \det\, [J_{ij}] \tag{66}$$

The particular body force loads of interest are due to the weight, linear acceleration, and centrifugal acceleration. For the weight load, the mass density must be given as well as the direction consines for the downward direction, then:

$$\{\dot{f}_B^{ij}\} = \rho g \begin{Bmatrix} n_x \\ n_y \\ n_z \end{Bmatrix} \tag{67}$$

where g is the acceleration of gravity. For linear acceleration, the three components of acceleration are needed; thus;

$$\{\dot{f}_B^{ij}\}_{acc} = -\rho \begin{Bmatrix} a_x \\ a_y \\ a_z \end{Bmatrix} \tag{68}$$

For centrifugal loading, the three components of the angular velocity vector, plus a point A on the axis of revolution must be given. Defining:

$$[\Omega] = \begin{bmatrix} 0 & -W_z & W_y \\ W_z & 0 & -W_x \\ -W_y & W_x & 0 \end{bmatrix} \tag{69}$$

then:

$$\{\dot{f}_B^{ij}\}_{cig} = [\Omega]^2 \begin{Bmatrix} x_{ij} - x_A \\ y_{ij} - y_A \\ z_{ij} - z_A \end{Bmatrix} \tag{70}$$

The traction loading for this element is generally divided into two types: pressure loading on the upper or lower surfaces and line loading along the element edges. Pressure loading performs normal to the surface (in the v_3 direction). The (x, y, z) components of the traction vector at integration point i(i = 1,9; top or bottom surface) is:

$$\vec{\rho}i = \rho i \begin{Bmatrix} v_{3,1}^{i} \\ v_{3,2}^{i} \\ v_{3,3}^{i} \end{Bmatrix} \tag{71}$$

where ρi is the magnitude of the pressure at node i. The 6x1 subvector of:

$$\left\{\dot{F}^T\right\} \text{pres}, \quad \left\{\dot{F}_i^T\right\} \quad \text{pres}$$

is thus:

$$\left\{\dot{F}_i^T\right\} \text{pres} = \beta i \; [T_i]^T \; [Q_{ij}]^T \; \vec{\rho}i \; | \; \vec{e}_{\xi}^{\,ij} \times \vec{e}_{\eta}^{\,ij} | \tag{72}$$

where

$$\beta i = \int_{-1}^{1} \int_{-1}^{1} Ni \; (\xi, \; \eta) \; d\xi d\eta \tag{73}$$

For an edge load, it is first assumed that the load is applied along the line $\zeta = 0$. For any edge, let the s direction be directed along an edge and let the n direction be normal to it. If the edge load at node E_j (where the three nodal points on the edge are labeled E_1, E_2, E_3), is given in terms of Cartesian components as $\dot{\vec{W}}_{ej}$, then the 6x1 equivalent subload vector $\{\dot{F}^T_{Ej}\}$ is given as:

$$\left\{\dot{F}^T_{Ej}\right\}_{line} = \gamma_{Ej}\ [T_{Ej}]^T\ [Q_{Ej,o}]^T\ \dot{\vec{W}}_{Ej}\ |\vec{e}^{\,Ej}_{so}| \tag{74}$$

Note that s corresponds to ζ or η depending on the edge. Sometimes, an edge load will be given in terms of normal and shear components. For shear loading in the s direction:

$$\dot{\vec{W}}_{Ej} = \dot{W}^{Ej}_{ns}\ \vec{e}^{\,Ej}_{so}/|\vec{e}^{\,Ej}_{so}| \tag{75}$$

For shear loading in the normal (ζ) direction:

$$\dot{\vec{W}}_{Ej} = \dot{W}^{Ej}_{n\zeta}\ \hat{V}_3^{\,Ej} \tag{76}$$

For normal loading:

$$\dot{\vec{W}}_{Ej} = \dot{W}^{Ej}_{nn}\ (\vec{e}^{\,Ej}_{so} \times \hat{V}_3)/|\vec{e}^{\,Ej}_{so} \times \vec{V}_3| \tag{77}$$

Other loads can be modeled as necessary in similar fashion. It hould be noted that "thermal loading" is incorporated into the constitutive law by the $\dot{\tau}$ term. In fact, $\dot{\tau}$ may be portioned into sum of terms. Each of these are then manipulated in an identical fashion.

2.2.3 20-NODED ISOPARAMETRIC SOLID ELEMENTS

The isoparametric solid elements permit the modeling of any general three-dimensional (3-D) object, since the elements represent a discretization of the object into finite elements which are 3-D continuous representations. The basic term "isoparametric" means that the elements utilize the same interpolating functions (also called "shape functions") to interpolate geometry, displacements, strains, and temperatures. It is, therefore, important that the user be aware that not any displacement, geometry, and temperature field to be analyzed is necessarily compatible with a given element mesh. This is particularly true where high temperature or strain gradients occur. The following sections discuss the basic element formulation assumptions and define the node numbering order and face definitions for pressure load input.

Shape functions are used to describe the variation of some function G within an element in terms of the nodal point values.

$$G(x,y,z) = \sum_{i=1}^{n} N_i G_i(x_i,y_i,z_i) \tag{78}$$

where

$G(x,y,z)$	=	the value of the function (such as displacement, temperature) at any point with coordinates (x,y,z) within an element
$G_i(x_i,z_i$	=	the value of the function at node point i
H_i	=	the element "shape function"
n	=	the number of nodes describing intraelement variation.

In order to ensure nonotonic convergence to the correct results, shape functions must satisfy several requirements. Satisfaction of these requirements results in convergence from an upper bound. These displacement function requirements are:

- They must include all possible rigid body displacements
- They must be able to represent constant strain states
- They must be differentiable within elements and compatible between adjacent elements.

While the above conditions prove valuable for establishing upper bounds for solutions, they are not essential. Incompatible displacement modes are widely and successfully used. Their principal disadvantage is that stiffness may no longer be bounded from above.

Curvilinear coordinates are introduced into the isoparametric concept to overcome the difficulty of formulating shape functions in global Cartesian coordinates. Also, generality in element geometry definition is obtained by this process.

A local curvilinear coordinate system (ξ,η,ζ) which ranges from -1 to 1 within each element is introduced in which shape functions are formulated. Also, a mapping from curvilinear to global coordinates is defined. A typical two dimensional element is shown in Figure 28.

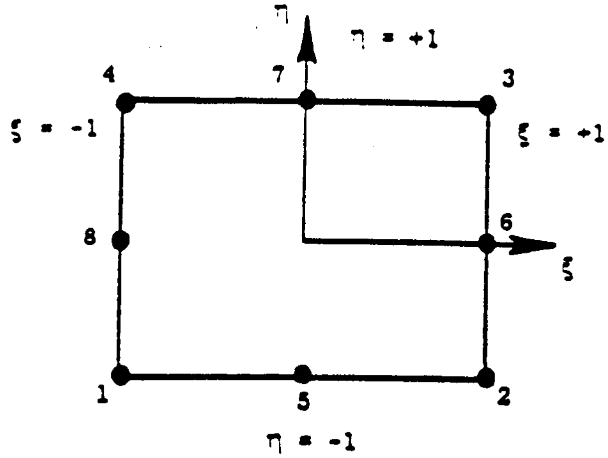
The same polynomial terms used in the Cartesian coordinates are used but with the curvilinear coordinates ξ,η,ζ replacing x, y, and z to generate shape functions. The ξ,η and ζ coordinates are the same for all global element configurations.

Structural finite element equilibrium equations:

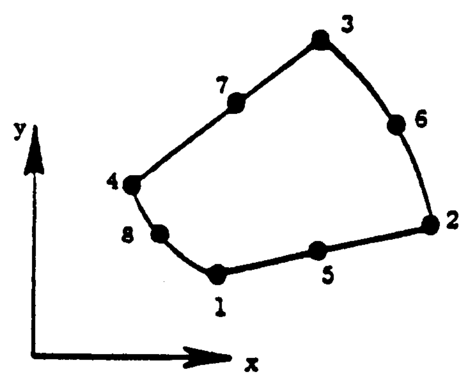
$$[M]\{\ddot{u}_i\} + [C]\{\dot{u}_i\} + [K]\{u_i\} = \{F_B\} + \{F_S\} + \{F_I\} + \{F_C\} + \{F_{NL}\} \quad (79)$$

$[M] = \int_v \rho[H]^T[H]dv$ "Consistent" mass matrix

$[C] =$ Damping matrix



(a) Curvilinear Coordinates



(b) Global Coordinates

Figure 28. Typical Two-Dimensional Element.

$[K] = \int_V [B]^T [D][B] dv$ Stiffness matrix

$\{F_B\} = \int_V [H]^T \{f_B\} dv$ Body forces

$\{F_S\} = \int_S [H^S]^T \{f_S\} dS$ Surface tractions

$\{F_I\} = \int_V [B]^T [D] \{\epsilon_T\} dv$ Initial strains

$\{F_{NL}\} = \int_V [B]^T [D] \{\epsilon_{NL}\} dv$ Nonlinear strains

where

$\{ui\}T = [u_1\ v_1\ w_1\ u_2\ v_2\ w_2 \ldots]$

$\{u\}T = [u\ v\ w]$

$\{u\} = [H]\{u_i\}$

$\{\epsilon\} = [B]\{u_i\}$

$\{\sigma\} = [D]\{\epsilon^e\} = [D](\{\epsilon^{TOT}\} - \{\epsilon^{THERM}\} - \{\epsilon_{NL}\})$

$\{f_B\}T = [f_{Bx}\ f_{By}\ f_{Bz}]$

$\{f_S\}T = [f_{Sx}\ f_{Sy}\ f_{Sz}]$

This element has been formulated with variable temperature general orthotropic material properties. During numerical integration for stiffness and equivalent nodal forces due to thermals, plasticity, and creep, the material properties at each integration point are evaluated at the temperature of that integration point. A Gauss integration scheme is used, and the user may choose an integration order of 2, 3, or 4 points in each direction (ξ, η, ζ).

The 20-noded solid has three displacement degrees of freedom on each of the 20 nodes for a total of 60 element degrees of freedom. Since this element utilizes a higher order displacement (and thus higher order strain) function, it can be used to model larger regions with fewer elements. The displacement functions yield a displacement distribution which is parabolic on an edge (three nodal points per edge).

The node numbering sequence for this element is shown in Figure 29. The user may define the location of Nodes N_1 and N_9 as desired, thus establishing the element pressure coordinate system and the resulting face numbering.

The interpolating functions can be defined starting with the basic corner noded shape functions:

$$G_1 = (1 + \xi)(1 + \eta)(1 + \zeta)/8 \qquad G_9 = (1 + \xi)(1 + \eta)(1 - \zeta)/8$$
$$G = (1 - \xi)(1 + \eta)(1 + \zeta)/8 \qquad G_{11} = (1 - \xi)(1 + \eta)(1 - \zeta)/8$$
$$G_5 = (1 - \xi)(1 - \eta)(1 + \zeta)/8 \qquad G_{13} = (1 - \xi)(1 - \eta)(1 - \zeta)/8$$
$$G_7 = (1 + \xi)(1 - \eta)(1 + \zeta)/8 \qquad G_{15} = (1 + \xi)(1 - \eta)(1 - \zeta)/8$$

The midside node shape functions are:

$$H_2 = (1 - \xi^2)(1 + \eta)(1 + \zeta)/4$$
$$H_4 = (1 - \xi)(1 - \eta^2)(1 + \zeta)/4$$
$$H_6 = (1 - \xi^2)(1 - \eta)(1 + \zeta)/4$$
$$H_8 = (1 + \xi)(1 - \eta^2)(1 + \zeta)/4$$
$$H_{10} = (1 - \xi^2)(1 + \eta)(1 - \zeta)/4$$
$$H_{12} = (1 - \xi)(1 - \eta^2)(1 - \zeta)/4$$
$$H_{14} = (1 - \xi^2)(1 - \eta)(1 - \zeta)/4$$
$$H_{16} = (1 + \xi)(1 - \eta^2)(1 - \zeta)/4$$
$$H_{17} = (1 + \xi)(1 + \eta)(1 - \zeta^2)/4$$
$$H_{18} = (1 - \xi)(1 + \eta)(1 - \zeta^2)/4$$
$$H_{19} = (1 - \xi)(1 - \eta)(1 - \zeta^2)/4$$
$$H_{20} = (1 + \xi)(1 - \eta)(1 - \zeta^2)/4$$

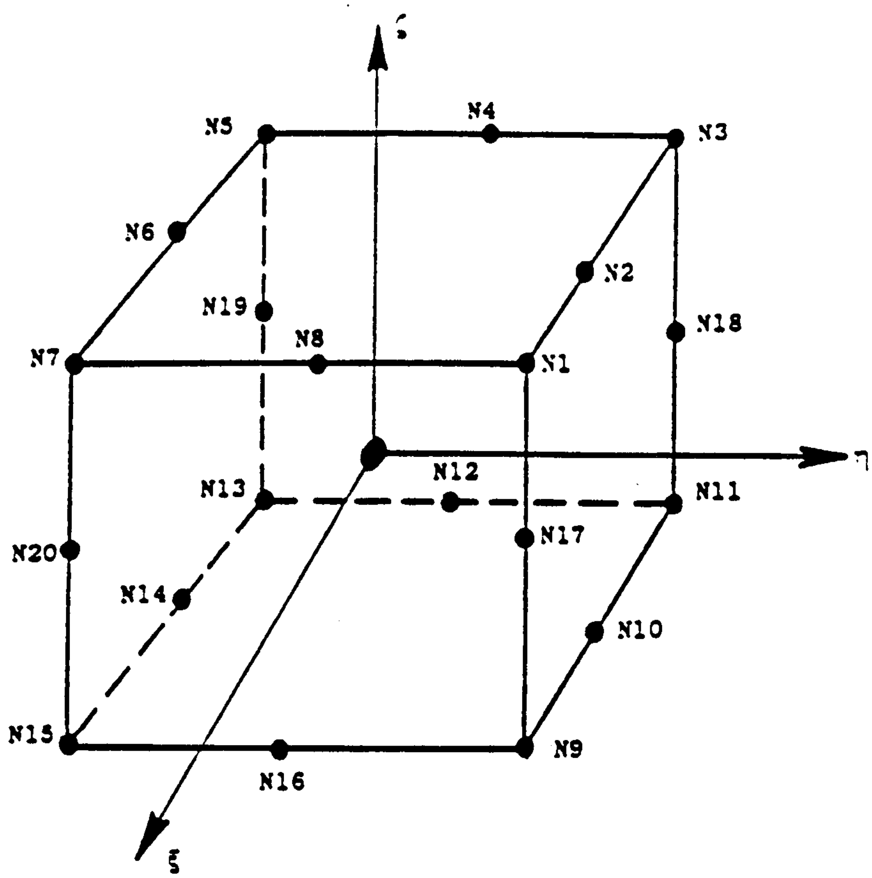


Figure 29. 20-Noded Solid Coordinate and Node Numbering System.

The modified corner node shape functions are:

$$
\begin{aligned}
H_1 &= G_1 - (H_2 + H_8 + H_{17})/2 \\
H_3 &= G_3 - (H_2 + H_4 + H_{18})/2 \\
H_5 &= G_5 - (H_4 + H_6 + H_{19})/2 \\
H_7 &= G_7 - (H_6 + H_8 + H_{20})/2 \\
H_9 &= G_9 - (H_{10} + H_{16} + H_{17})/2 \\
H_{11} &= G_{11} - (H_{10} + H_{12} + H_{18})/2 \\
H_{13} &= G_{13} - (H_{12} + H_{14} + H_{19})/2 \\
H_{15} &= G_{17} - (H_{14} + H_{16} + H_{20})/2
\end{aligned}
$$

This element requires no incompatible modes to simulate bending properly, since the displacement functions are complete quadratics. Thus the displacements (and strains and temperatures) are computed as:

$$u = \sum_{i=1}^{20} H_i U_i$$

$$v = \sum_{i=1}^{20} H_i V_i$$

$$w = \sum_{i=1}^{20} H_i W_i$$

Given the coordinate system (ξ, η, ζ) as previously established, we can also now define the face numbering conventions and order of nodes on a face. These definitions are needed to established conventions for inputting pressure levels on the element and numbering of faces when displaying surface stresses on the faces. These conventions are summarized below.

Face No.	Location	Nodes and Node Order on Face							
1	$\xi = +1$	N1	N8	N7	N20	N15	N16	N9	N17
2	$\eta = +1$	N1	N17	N9	N10	N11	N18	N3	N2
3	$\zeta = +1$	N1	N2	N3	N4	N4	N6	N7	N8
4	$\xi = -1$	N13	N19	N5	N4	N3	N18	N11	N12
5	$\eta = -1$	N13	N14	N15	N20	N7	N6	N5	N19
6	$\zeta = -1$	N13	N12	N11	N10	N9	N16	N15	N14

This element has been formulated with variable temperature and general orthotropic material properties. During numerical integration for stiffness and equivalent nodal forces due to thermals, plasticity, and creep, the material properties at each integration point are evaluated at the temperature of that integration point. A Gauss integration scheme is used, and the user may choose an integration order of 2, 3, or 4 points in each direction (ξ,η,ζ).

2.3 COMPUTER PROGRAMS

A separate computer program has been developed for each combination of constitutive model-formulation model. Each program provides a functional, standalone capability for performing cyclic nonlinear structural analysis. In addition, the analysis capabilities incorporated into each program can be abstracted in subroutine form for incorporation into other codes or to form new combinations. These programs will provide the structural analyst with a matrix of capabilities involving the constitutive model-formulation models from which he will be able to select the combination that satisfies his particular needs.

The total amount of software developed for this contract is large. It will be presented in separate manuals as required. To illustrate the program architecture, Appendix B is a Fortran source listing for a main calling program. Appendix C contains a listing of the input for the various programs. Appendix D contains the listings of the DATA DECK GENERATORS programs. These programs can be used to interactively generate the input for the structural codes.

2.3.1 PROGRAM ARCHITECTURE

The program architecture employs state-of-the-art techniques to maximize efficiency, utility, and portability. Among these features are the following:

(i) User Friendly I/O

- Free format data input
- Global, local coordinate system, (Cartesian, Cylindrical, Spherical)
- Automatic generation of nodal and elemental attributes
- User-controlled optional print out

  Nodal Displacements

  Nodal Forces

  Element Forces

  Element Stresses and Strains

(ii) Programming Efficiency

- Dynamic core allocation
- Optimization of file/core utilization
- Blocked column skyline equation solver

(iii) Accurate and Economical Solution Techniques

- Right-hand side pseudoforce technique
- Accelerators for the iteration scheme
- Convergence criteria based on both the local plastic strain and the global displacements.

Figure 30 is a generic flow chart for these programs.

2.3.2 LINEAR VARIATION OF LOADS

The ability to model piecewise linear load histories was also included in the finite element code. This capability is particularly useful when modeling stress strain tests or fatigue loops, and also for certain analysis applications. Since the inelastic strain rate could be expected to change dramatically during a linear load history, it is important to include a dynamic time-incrementing procedure. The term "load case" is used to denote a time period for which the initial and boundary conditions are defined and vary linearly between the end points.

In order to incorporate linear load histories into this scheme, the total displacement vector is decomposed into elastic and inelastic components as:

$$\{d^T\} = \{d^E\} + \{d^I\}. \tag{80}$$

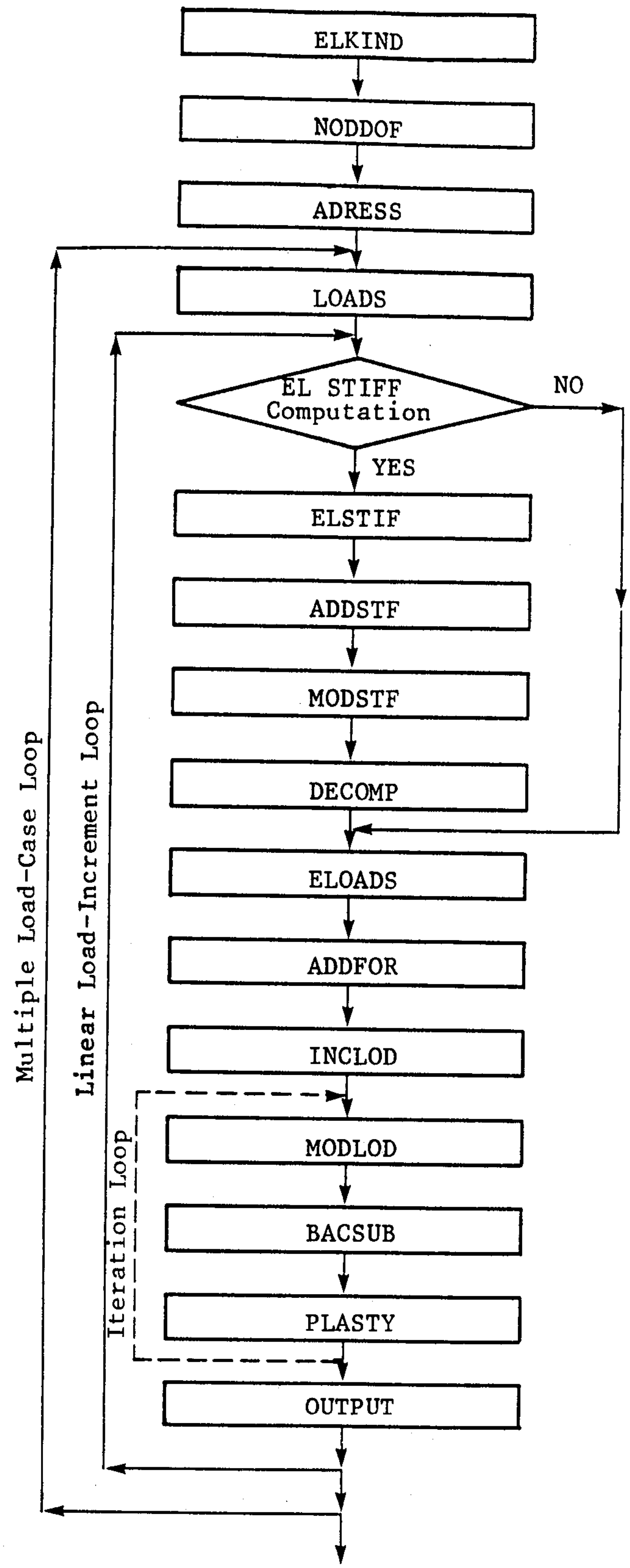


Figure 30. Generic Flow Chart.

The vector $\{d^E\}$ is the displacement due to applied thermomechanical forces, and $\{d^I\}$ is the displacement due to the inelastic pseudoforces. These displacementt vectors can be calculated using:

$$\{d^E\} = [K]^{-1} \{F\}, \tag{81}$$

and:

$$\{d^I\} = [K]^{-1} \{F^I\}. \tag{82}$$

The elastic displacement are obtained for the initial and final thermomechanical loads in the load case. The elastic displacements at any time in the load case are given by:

$$\left\{d^E\right\} = \left\{d^E\right\}_o + \left(\frac{t - t_o}{t_f - t_o}\right) \left(\left\{d^E\right\}_f - \left\{d^E\right\}_o\right) \tag{83}$$

The vectors $\{d^E\}_o$ and $\{d^E\}_f$ are the elastic displacements due to the initial and final applied thermomechanical forces. The current time in the load case is t, and t_o and t_f are the initial and final times in the load case.

The displacements due to the inelastic strains at any time during the load case are given by:

$$\left\{d^I\right\} = \left\{d^I\right\}_o + \sum_{\substack{\text{Time} \\ \text{Increments}}} \left\{\Delta d^I\right\} \tag{84}$$

The vector $\{d^I\}_o$ is the vector of displacements due to inelastic strains at the beginning of the load case, and $\{\Delta d^I\}$ is a displacement increment due to the inelastic strains during a time step.
The increment in displacements $\{\Delta d^I\}$ due to the change in inelastic strains $\{\Delta\varepsilon^I\}$ during a time step is computed using:

$$\left\{\Delta d^I\right\} = [K]^{-1}\left\{\Delta F^I\right\} \tag{85}$$

The inelastic pseudoforce increment $\{\Delta F^I\}$ is calculated from:

$$\left\{\Delta F^I\right\} = \sum_{\text{Elements}} \int [B]^T [E] \left\{\Delta\varepsilon^I\right\} dV \tag{86}$$

where $\{\Delta\varepsilon^I\}$ is the change in inelastic strain during the time increment.

At the beginning of a load case, the initial and final elastic displacements are computed using equation (81) , and the displacements due to prior inelastic strains are computed using equation (82). The total strains at the beginning of the load case are recovered for each integration point and the elastic strains are computed from:

$$\left\{\varepsilon^E\right\} = \left\{\varepsilon^T\right\} - \left\{\varepsilon^{\alpha\Delta T}\right\} - \left\{\varepsilon^I\right\} \tag{87}$$

where $\{\varepsilon^T\}$ are the total strains and $\{\varepsilon^{\alpha\Delta T}\}$ are the thermal strains.

Using the current values of the state variables and the stress state variable evolution rates. Before entering the time loop, an initial time increment is computed and the inelastic strain increments are estimated using a forward Euler integration formula. From the estimated inelastic strain increments, an initial estimate is made for the inelastic pseudoforce increment using Equation (86).

The usual technique employed with the initial strain method is to assume that the incremental inelastic force $\{\Delta F^I\}$, the corresponding displacements $\{\Delta d^I\}$, and the inelastic strain increments $\{\Delta \varepsilon^I\}$, are all zero on the first iteration of a time step. The stability of the method can be improved considerably when a forward Euler integration of the inelastic strain rates is used to make an estimate of $\{\Delta \varepsilon^I\}$, $\{\Delta F^J\}$ and $\{\Delta d^I\}$ on the first iteration. This method results in an initial estimate which is much closer to the solution. In sample cases, the overall number of iterations was reduced by more than one half.

The procedure during a time increment is to estimate the solution on the first iteration using a forward Euler scheme as outlined above. Then displacements, strains, stresses, inelastic strain rates, and state variable evolution rates are computed at the end of the time increment. The inelastic strains and state variables are integrated over the time increment and an improved inelastic force is computed. The procedure is repeated until convergence is achieved at the end of the time increment. Figure 31 summarizes the logic.

2.3.3 DYNAMIC TIME INCREMENTING

In a computer code that allows a linear variation of loads, a dynamic time incrementing scheme is very desirable since large excursions in stress and inelastic strain rate are to be expected. The procedure used to compute the time increments requires a certain amount of initial experimentation to determine appropriate time step control parameters. However, once this has been done, the procedure works quite well and is a tremendous improvement in economy over a constant time increment.

Three separate time step control criteria are used. These are the maximum stress increment, maximum inelastic strain increment, and maximum rate of change of the inelastic strain rate. The minimum time step calculated from the three criteria is the value actually used. Since the calculations are based on values readily available from the previous time step, little computational effort is required.

2.3.4 STRESS INCREMENT CRITERION

A maximum stress increment criterion is used to control the time increment during primarily elastic excursions. This criterion is necessary to prevent overshoot of the point where significant inelastic strain rates begin. The calculation for the time increment is given by:

$$\Delta t_k = \Delta t_{k-1} \left[\frac{\Delta\sigma_{INC}}{\left(\Delta\sigma_{K-1}\right)_{max}} \right] \tag{88}$$

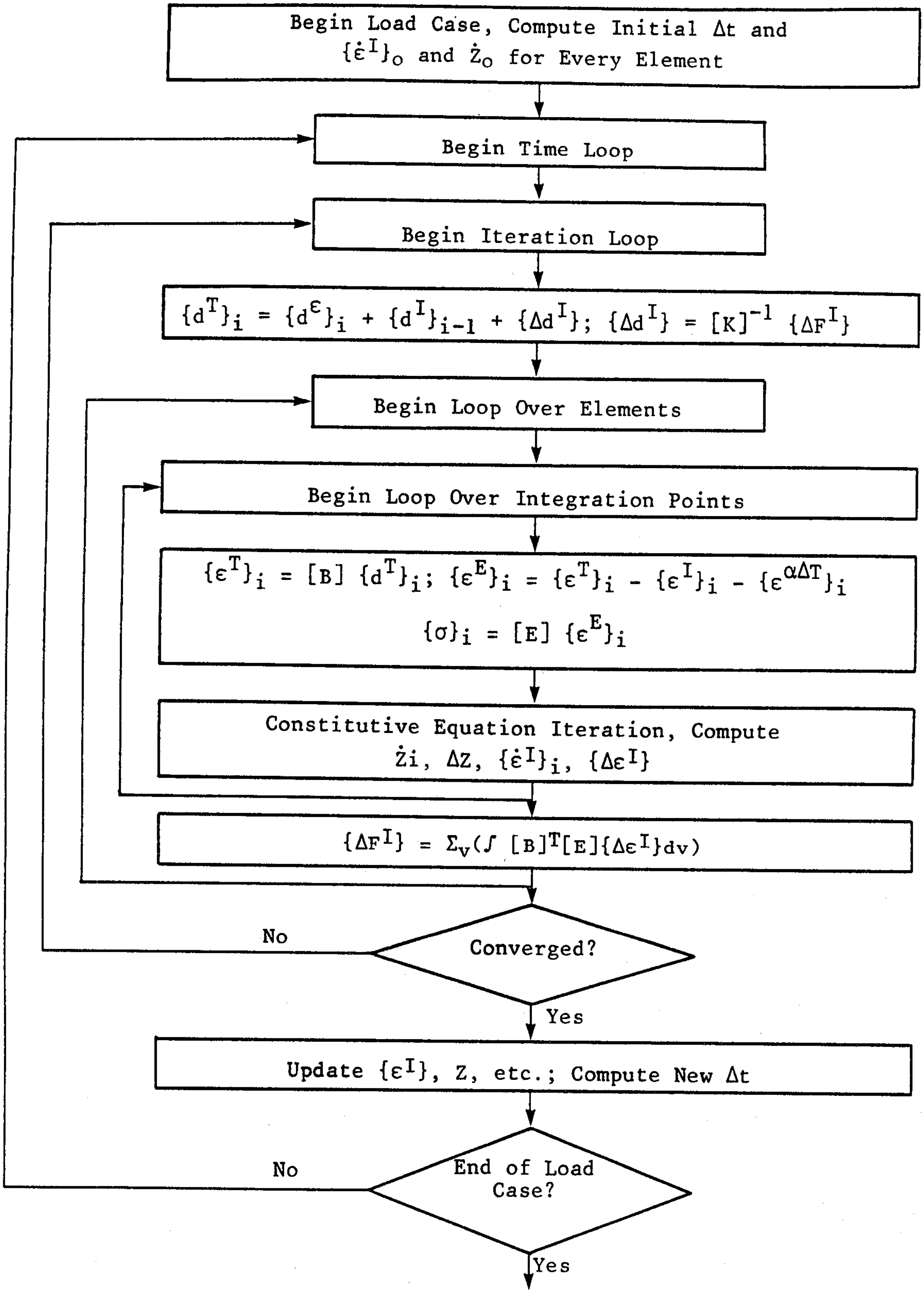


Figure 31. Flow Chart of Finite Element Solution Procedure.

where Δt_{R-1} is the previous time increment, $(\Delta\sigma_{k-1})_{max}$ is the maximum change in effect stress for all integration points during the previous time increment, and $\Delta\sigma_{INC}$ is the maximum desired stress increment. The value for $\Delta\sigma_{INC}$ is program input and will vary somewhat, depending on material constants. Typical values are about 15 MPa.

2.3.5 INELASTIC STRAIN INCREMENT CRITERION

The inelastic strain increment criterion controls the time step when the stress and inelastic strain rates are not changing significantly. This is given by:

$$\Delta t_k = \Delta t_{k-1} \left[\frac{\Delta\varepsilon^I_{INC}}{\Delta\varepsilon^I_{K-1\ max}} \right] \tag{89}$$

The maximum change in effective inelastic strain for all integration points during the previous time increment is $(\Delta\varepsilon^I_{k-1\ max})$. $\Delta\varepsilon^I_{INC}$ is the maximum desired inelastic strain increment. The value for $\Delta\varepsilon^I_{INC}$ is program input, and typical valus are about 0.000100.

2.3.6 RATE OF CHANGE OF THE INELASTIC STRAIN RATE CRITERION

The $\dot{\varepsilon}^I$ criterion controls the time increment when the inelastic strain rate is changing rapidly, such as in the "knee" of a stress strain curve. The quantity $\ddot{\varepsilon}^I$ is a measure of how close the initial forward Euler estimation is to the final converged solution. The backward difference formula:

$$\ddot{\varepsilon}_i^I \simeq \left(\dot{\varepsilon}_{i-1}^I - \dot{\varepsilon}_{i-2}^I\right)/\Delta t_{k-1} \tag{90}$$

is used to estimate $\ddot{\varepsilon}^I{}_i$. The maximum value of $\ddot{\varepsilon}^I$ for all integration points, $\ddot{\varepsilon}^I{}_{max}$ is used to estimate the next time step using:

$$\Delta t_K = \sqrt{\frac{2\Delta t_{k-1}\Delta\varepsilon_{INC}^I\, e}{\left(\ddot{\varepsilon}_i^I\right)_{max}}} \tag{91}$$

The parameter e is the maximum desired percent error by which the initial forward Euler estimation is in error. The value for e is program input and typical values are about 0.01. Equation (91) is derived simply from taking the difference between a Euler integration scheme and the more accurate second order Adams-Moulton method.

APPENDIX A

RESULTS OF LITERATURE SURVEY

APPENDIX A REFERENCES

1. Padovan, J. and Tovichakchaikul, S., "Self-Adaptive Predictor-Corrector Algorithms for Static Non-Linear Structural Analysis," Computer Structures 15, pp. 365-377, 1982.

2. Padovan, J. and Tovichakchaikul, S., "On the Solution of Creep Induced Buckling in General Structure," Computer Structures 15, pp. 379-392, 1982.

3. Liu, W.K., and Belytschko, T., "Mixed-Time Implicit-Explicit Finite Elements for Transient Analysis," Computer Structures 15, pp. 445-450, 1982.

4. Zhang, L. and Owen, D.R.J., "A Modified Secant Newton Method for Non-Linear Problems," Computer Structures 15, pp. 543-547, 1982.

5. Zienkiewicz, O.C., and Parekh, C.J., "Transient Field Problems: Two-Dimensional and Three-Dimensional Analysis by Isoparametric Finite Elements," Int. J. Num. Metho. Engrg. 2, pp. 61-71, 1971.

6. Stricklin, J.A., and Haisler, W.E., "Formulations and Solution Procedures for Nonlinear Structural Analysis," Computer Structures 7, pp. 125-136, 1977.

7. Barsoum, R.S., "A Convergent Method for Cyclic Plasticity Analysis with Application to Nuclear Components," Int. J. Num. Meth. Engr. 6, pp. 227-236, 1973.

8. Powell, G. and Simons, J., "Improved Iteration Strategy for Nonlinear Structures," Int. J. Num. Meth. Engrg. 17, pp. 1455-1467, 1981.

9. Levy, A. and Pifko, A.B., "On Computational Strategies for Problems Involving Plasticity and Creep," Int. J. Num. Meth. Engrg. 17, pp. 747-771, 1981.

10. Tsybenko, A.S., Vashchenko, N.G., Krishchuk, N.G., and Kulakooskii, V.N., "Automatic Formulation of Triangular - Element Grid for Arbitrary Plane Regions," Translated from Problems Prochnosti, 12, pp. 84-89, 1980.

11. Sobrezczansi-Sobieski, J. and Hajela, R., "Accuracy of an Approximate Static Structural Analysis Technique Based on Stiffness Matrix Eignmodels," AIAA Paper No. 79-0748, 1979.

12. Bergan, P.G., Horrigmoe, G., Krikeland, B. and Soreide, T.H., "Solution Techniques for Nonlinear Finite Element Problems," Int. J. Num. Meth. Engrg. 12, pp. 1677-1696, 1978.

13. Jirousek, J. and Teodorescu, P., "Large Finite Elements Method for the Solution of Problems in the Theory of Elasticity," Computer Structures 15, pp. 575-587, 1982.

14. Sorek, S. and Blech, J.J., "Finite-Element Technique for Solving Problems Formulated by Hamilton's Principle," Computer Structures 15, pp. 533-541, 1982.

15. Bausha, I. and Rheinboldt, W.C., "Adaptive Approaches and Reliability Estimations in Finite Element Analysis," Comput. Meth. Appl. Mech. Engrg. 17/18, pp. 519-540, 1979.

16. Boyle, E.T. and Jennings, A., "Accelerating the Convergence of Elastic - Plastic Stress Analysis," Int. J. Num. Meth. Engr. 7, pp. 232-235, 1973.

17. Morjaria, M., Sarihan, V., and Mukheree, S., "Comparison of Boundary Element and Finite Element Methods in Two-Dimensional Inelastic Analysis," DOE Report No. COO-2733-22, 1979.

18. Rzasnicki, W., Mendelson, A., Albers, L.U., and Raftoponlos, D.D., "Application of Boundary Integral Method to Elasto-Plastic Analysis of V-Notched Beams," NASA TN D-7637, 1974.

19. Katz, C. and Werner, H., "Implementation of Nonlinear Boundary Conditions in Finite Element Analysis," Computer Structures 15, pp. 299-304, 1982.

20. Tong, P., Maw, S.T., and Pian, T.H.H., "Derivation of Geometric Stiffness and Mass Matrices for Finite Element Hybrid Models," Int. J. Solids Structures 10, pp. 919-932, 1974.

21. Olderson, R.G., Tani, M.A., Tree, D.J., and Hill, R.J., "A Three-Dimensional Approach to the Optimization of a Gas Turbine Disc and Blade Attachment," AIAA Paper No. 75-1312, 1975.

22. Rice, J.R., "On the Structure of Stress-Strain Relations for Time-Dependent Plastic Deformation in Metals," Journal Appl. Mech., pp. 728-737, 1970.

23. Ponter, A.R.S. and Leckie, F.A., "Constitutive Relationships for the Time-Dependent Deformation of Metals," J. Engrg. Mat. Tech., pp. 47-51, 1976.

24. Gittus, J.H., "Dislocation - Creep Under Cyclic Stressing: Physical Model and Theoretical Equations," ACTA Metal. 26, pp. 305-317, 1978.

25. Allen, J.M., "Effect of Temperature Dependent Mechanical Properties on Thermal Stress in Cooled Turbine Blades," Journal Engrg. Power 1-5, 1981.

26. Kaufman, A. and Gaugler, R., "Cyclic Structural Analyses of Air-Cooled Gas Turbine Blades and Vanes," SAE Paper No. 760918, 1976.

27. Peterson, M.R., Alderson, R.G., Stockton, R.J., and Tree, D.J., "Three-Dimensional Finite-Element Techniques for Gas Turbine Blade Life Prediction."

28. Allen, J.M. and Erickson, L.B., "NASTRAN Analysis of a Turbine Blade and Comparison with Test and Field Data," ASME Paper No. 75-GT-77, 1975.

29. Wolf, D.S., "Stress Analysis of a First Turbine Vane Using a Three-Dimensional Model with Nonlinear Material Behavior Subjected to Transient Temperatures," AIAA 81-1437, 1981.

30. Jhansale, H.R. and Sharma, G.K., "Inelastic Behavior of Structural Metals Under Complex Cyclic Loadings," AIAA/ASME Paper No. 77-366, 1977.

31. Iwan, W.D., "On a Class of Models for the Yielding Behavior for Continuous and Composite Systems," J. Appl. Mech., pp. 612-617, 1917.

32. Drucker, D.C. and Palgen, L., "On Stress-Strain Relations Suitable for Cyclic and Other Loading," J. Appl. Mech. 48, pp. 479-485, 1981.

33. Besseling, J.F., "A Theory of Plastic Flow for Anisotropic Hardening in Plastic Deformation of an Initially Isotropic Material," Nat. Aero. Res. Inst. Report 5.410, 1953.

34. Mroz, Z., "An Attempt to Describe the Behavior for Metals Under Cyclic Loads Using a More General Work-Hardening Model," ACTA Mechanica 7, pp. 199-212, 1969.

35. DeKoning, A.U., "A Continuum Theory of Time Independent Plasticity Based on a Volume Fractions Model," NLR The Netherlands, NLR MP 77017 U, 1978.

36. Williams, J.F., and Svensson, N.L., "A Rationally Based Yield Criterion for Work-Hardening Materials," ACTA Mechanica 6, pp. 104-114, 1971.

37. Baladi, G.Y., "Numerical Implementation of a Transverse-Isotropic Inelastic, Work-Hardening Constitutive Model," Nucl. Engrg. Design 46, pp. 263-272, 1978.

38. Wells, C.H., "An Analysis of the Bauschinger Effect in Some Engineering Alloys," ASME Paper No. 47-WA, Met-1, 1967.

39. Radomski, M. and White, D.J., "Some Theoretical Considerations Relating to Strain Concentration in Elastic-Plastic Bending of Beams," J. Strain Analysis 3, pp. 304-312, 1968.

40. White, D.J. and Radomski, M., "Strain Concentration in Beams Under Cyclic Plastic Straining," J. Strain Analysis 3, pp. 313-324, 1968.

41. Malinin, N.N. and Niquin, A.A., "The Influence of Stress Concentration on Creep Rupture at Non-Stationary Loading," Int. J. Mech. Sci. 19, pp. 521-531, 1977.

42. Kiselev, V.A., and Semishkin, V.P., "Stress State Kinetics Near Stress Raisers Under High-Temperature Creep Conditions," Problemy Prochnostic 2, pp. 10-13, 1981.

43. Goldman, A. Ya and Tsygavkoo, S.A., "Model of a Physically Nonlinear Viscoelastic Medium for Describing the Creep of Polymers in a Complex Stress State," Problemy Prochnosti 2, pp. 14-18, 1981.

44. Bodner, S.R., Partom, I., and Partom, Y., "Uniaxial Cyclic Loading of Elastic-Viscoplastic Material," ASME J. Appl. Mech., Vol. 46, p. 805, 1979.

45. Stouffer, D.C. and Bodner, S.R., "A Constitutive Model for the Deformation Induced Anisotropic Plastic Flow of Metals," Int. J. Engrg. Sci., Vol. 17, p. 737, 1979.

46. Bodner, S.R. and Partom, Y., "A Large Deformation Elastic-Viscoplastic Analysis of a Thick-Walled Spherical Shell," ASME Journal of Applied Mechanics, September 1972.

47. Stouffer, D.C. and Bodner, S.R., "A Relationship Between Theory and Experiment for a State Variable Constitutive Equation," Air Force Materials Laboratory, AFWAL-TR-80-4194, 1981.

48. Bodner, S.R., "Representation of Time Dependent Mechanical Behavior of René 95 by Constitutive Equations," Air Force Materials Laboratory, AFML-TX-79-4116, 1979.

49. Bodner, S.R. and Lindholm, U.S., "An Incremental Criterion for Time-Dependent Failure of Materials," ASME Journal of Engineering Materials and Technology, April 1976.

50. Merzer, A.M., "Steady and Transient Creep Behavior Based on Unified Constitutive Equation," ASME Journal of Engineering Materials and Technology, January 1982.

51. Bodner, S.R. and Partom, Y., "Constitutive Equations for Elastic Viscoplastic Strain-Hardening Materials," ASME Journal of Applied Mechanics, June 1975.

52. Abuelfoutouh, N.M., "A Thermodynamically Consistent Constitutive Model for Inelastic Flow of Materials," PhD Dissertation, University of Cincinnati, 1983.

53. Haisler, W.E. and Cronenworth, J., "An Uncoupled Viscoplastic Constitutive Model for Metals at Elevated Temperatures," presented at AIAA/ASME SSDM meeting, Lake Tahoe, May 2-4, 1983.

54. Bradley W.L. and Yuen, S., "A New Uncoupled Viscoplastic Constitutive Model," presented at Symposium on Nonlinear Constitutive Relations for High Temperature Applications, The University of Akron, Akron, Ohio, May 19-20, 1982.

55. Hasiler, W.E. and Sanders, D.R., "Elastic-Plastic-Creep-Large Strain Analysis at Elevated Temperature by the Finite Element Method," Computers and Structures, Vol. 10, pp. 375-381, 1979.

56. Allen, D.H., and Haisler, W.E., "A Theory for Analysis of Thermoplastic Materials," Computers and Structures, Vol. 13, pp. 129-135, 1981.

57. Kreig, R.D., Swearingen, J.C., and Rohde, R.W., "A Physically-Based Internal Variable Model for Rate-Dependent Plasticity," Proceedings ASME/CSME DVP Conference, pp. 15-27, 1978.

58. Bathe, K.J., "Finite Element Procedures in Engineering Analysis," Prentice-Hall, Englewood Cliffs, New Jersey, 1982.

59. Oytana, C., Delobelle, P. and Mermet, A., "Constitutive Equations Study in Biaxial Stress Experiments," ASME Journal of Engineering Materials and Technology, January 1982.

60. Williams, J. and Cocks, A., "Reference Stress and Temperature for Nonisothermal Creep of Structures," ASME Journal of Applied Mechanics, December 1979.

61. Wire, G.L., et. al., "A State Variable Analysis of Inelastic Deformation of Thin Walled Tubes," ASME Journal of Engineering Materials and Technology, October 1981.

62. Jackson, M.S., et. al., "A Phenomenological Model for Transient Deformation Based on State Variables," ASME Journal of Engineering Materials and Technology, October 1981.

63. Sawzuk, A. and Tramprzynski, W., "A Theory of Anisotropic Creep After Plastic Pre-Straining," Int. J. Mech. Sci, Vol. 24, No. 11, 1982.

64. Weng, G.J., "A Unified Self Consistent Theory for the Plastic-Creep Deformation of Metals," ASME Journal of Applied Mechanics, December 1982.

65. Ohno, N., "A Constitutive Model of Cyclic Plasticity with a Nonhardening Strain Region," ASME Journal of Applied Mechanics, December 1982.

66. Bundiansky, B. and Wu, T.T., "Theoretical Prediction of Plastic Strains of Polycrystals," Proc. of the Fourth U.S. Nat. Congress of Applied Mechanics, Vol. II, ASME, pp. 1175-1185, 1962.

67. Lin, T.H. and Ito, M., "Theoretical Plastic Stress-Strain Relationship of a Polycrystal and the Comparisons with the Von Mises and the Tresca Plasticity Theories," Int. J. Engrg. Sci., Vol, 4, pp. 543-561, 1966.

68. Laflen, J.H. and Jaske, C.E., "Cyclic Relaxation Response Under Creep-Fatigue Conditions," Stress Relaxation Testing, ASTM STP 676, Alfred Fox, Ed., American Society for Testing and Materials, pp. 182-206, 1979.

69. Thompson, A.W., "Polycrystal Hardening," Work Hardening in Tension and Fatigue, A.W. Thompson, Editor, The Metallurgical Society of ASME, 1977.

70. Valanis, K.C., "A Theory of Viscoplasticity Without a Yield Surface," Parts I and II, Archives of Mechanics, Archiwum, Mechaniki Stoswanej, 24, 4, 1971.

71. Pugh, C.E., "Constitutive Equations for Creep Analysis of Liquid Moderated Fast Breeder Reactor (LMFBR) Components," Advances in Design for Elevated Temperature Environment, S.Y. Zamrik and N.I. Jetter, Eds. ASME, 1975.

72. Krempl, E., "On the Interaction of Rate and History Dependence in Structural Metals," ACTA Mechanica, Vol. 22, pp. 53-90, 1975.

73. Krempl, E. and Hewelt, P., "The Constant Volume Hypothesis for the Inelastic Deformation of Metals in the Small Strain Range," Mechanics Research Communications, Vol. 7, No. 5, pp. 283-288, 1980.

74. Hart, E.W., Li, C.J., Yamada, H, and Wire, G.L., "Phenomenological Theory: A Guide to Constitutive Relations and Fundamental Deformation Properties," Constitutive Equations in Plasticity, The MIT Press, Cambridge, Mass., London, Argon, A.S., Ed, 1975, pp. 149-197.

75. Walker, K.P., "Research and Development Program for Nonlinear Structural Modeling with Advanced Time-Temperature Dependent Constitutive Relationship," NASA CR-165533, November 1981.

76. Laflen, J.H., "A Constitutive Theory for Metal Creep," PhD Dissertation, University of Cincinnati, 1976.

77. Hutchinson, J.W., "Bounds and Self-Consistent Estimates for Creep of Polycrystalline Materials," Proc. of R. Soc. Lond. A. 348, 1976, pp. 101-127.

78. Bishop, J.F.W. and Hill, R., "A Theory of the Plastic Distortion of a Polycrystalline Aggregate Under Combined Stresses," Phil. Mag., 42, pp. 414-427, 1951.

79. Bishop, J.F.W. and Hill, R., "A Theoretical Derivation of the Plastic Properties of a Polycrystalline Face-Centered Metal," Phil. Mag., 42, pp. 1298-1307, 1951.

80. Bishop, J.F.W., "A Theoretical Examination of the Plastic Deformation of Crystals by Glide," Phil, Mag., pp. 51-64, 1952.

81. Batdorf, S.B. and Bundiansky, B., "A Mathematical Theory of Plasticity Based on the Concept of Slip," NACA TN 1871, April 1949.

82. Brown, G.M., "A Self-Consistent Polycrystalline Model for Creep Under Combined Stress States," J. Mech. Phys. Solids, Vol. 18, pp. 367-381, 1970.

83. Patel, N.R., "An Analysis of Visco-Plastic Behavior of Metals," PhD Dissertation, Columbia University, University Microfilms, Ann Arbor, Michigan, 1977.

84. Culie, J.P., Cailletaud, G. and Lasalmonie, A., "Internal Stress in Viscoplasticity. Comparison Between Mechanical and Microscopic Approaches," La Recherche Aerospatiale, O.N.E.R.A., 1982-2.

85. Dyson, B.F., Verma, A.K. and Szkopiak, Z.C., "The Influence of Stress State on Creep Resistance: Experiments and Modeling," ACTA Metallurgica, Vol. 29, 1981.

86. Weng, G.J., "Self-Consistent Determination of Time-Dependent Behavior of Metals," ASME Journal of Applied Mechanics, March 1981.

87. Weng, G.J., "A Physically Consistent Method for the Prediction of Creep Behavior of Metals," ASME Journal of Applied Mechanics, December 1979.

88. Perrone, N. and Bhadra, P., "A Simplified Method to Account for Plastic Rate Sensitivity with Large Deformations," ASME Journal of Applied Mechanics, December 1979.

89. Argon, A.S. and Takeuchi, S., "Internal Stresses in Power-Law Creep," ACTA Metallurgica, Vol. 29, 1981.

90. Rubin, M., "A Thermoelastic-Viscoplastic Model with a Rate-Dependent Yield Strength," ASME Journal of Applied Mechanics, June 1982.

91. Mroz, Z., "Hardening and Degradation Rules for Metals Under Monotonic and Cyclic Loading," ASME Journal of Engineering Materials and Technology, April 1983.

92. Lamba, H.S. and Sidebottom, O.M., "Cyclic Plasticity for Nonproportional Paths," ASME Journal of Engineering Materials and Technology, January 1978.

93. Mukherjee, S. and Kumar, V., "Numerical Analysis of Time-Dependent Inelastic Deformation in Metallic Media Using the Boundary-Integral Equation Method," ASME, Journal of Applied Mechanics, December 1978.

94. Zhang, L. and Owne, D., "A Modified Secant Newton Method for Nonlinear Problems," Computer and Structures, Vol. 15, 1982.

95. Kamat, M., and Hayduk, R., "Energy Minimization Versus Pseudo Force Technique for Nonlinear Structural Analysis," Computers and Structures, Vol. 11, 1980.

96. Wellford, L. and Sen, H., "A Selective Relation Iterative Solution Technique for Nonlinear Structural Analysis Problems," International Journal for Numerical Methods in Engineering, Vol. 17, 1981.

97. Kumar, V., Morjaria, M. and Mukherjee, S., "Numerical Integration of Some Stiff Constitutive Models of Inelastic Deformation," ASME Journal of Engineering Materials and Technology, Vol. 120, January 1980.

98. Morjaria, M. and Mukherjee, S., "Improved Boundary - Integral Equation Method for Time-Dependent Inelastic Deformation in Metals," International Journal for Numerical Methods in Engineering, Vol. 15, pp. 97-111, 1980.

99. Kaneko, I. and Ha, C., "A Decomposition Procedure for Large-Scale Optimum Plastic Design Problems," International Journal for Numerical Methods in Engineering, Vol. 19, pp. 873-889, 1983.

100. Miller, A. and Shih, C., "An Improved Method for Numerical Integration of Constitutive Equations of the Work-Hardening-Recovery Type," ASME, Journal of Engineering Materials and Technology, July 1977.

101. Lubarda, V. and Lee, E., "A Correct Definition of Elastic and Plastic Deformation and its Computational Significance," ASME, Journal of Applied Mechanics, Vol. 48, March 1981.

102. Belytschko, T. and Schoeberle, D., "On the Unconditional Stability of an Implicit Algorithm for Nonlinear Structural Dynamics," ASME Journal of Applied Mechanics, December 1975.

103. Surana, K., "Geometrically Nonlinear Formulation for the Axisymmetric Shell Elements," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 477-502, 1982.

104. Barlow, J., "Constraint Relationships in Linear and Nonlinear Finite Element Analyses," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 521-533, 1982.

105. Cook, W., "A Finite Element Model for Nonlinear Shells of Revolution," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 135-149, 1982.

106. Dinis, L. and Owen, D., "Elasto-Viscoplastic and Elasto-Plastic Large Deformation Analysis of Thin Plates and Shells," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 591-607, 1982.

107. Franchi, A. and Cohn, M., "Strain-Softening and Large Displacement Analysis in Structural Plasticity," Computers and Structures, Vol. 11, pp. 421-427, 1980.

108. Pietruszozak, St. and Mroz, Z., "Finite Element Analysis of Deformation of Strain-Softening Materials," International Journal for Numerical Methods in Engineering, Vol. 17, pp. 327-334, 1981.

109. Morjaria, M. and Mukherjee, S., "Finite Element Analysis of Time-Dependent Inelastic Deformation in the Presence of Transient Thermal Stresses," International Journal for Numerical Methods in Engineering, Vol. 17, pp. 909-921, 1981.

110. Owen, D. and Figueiras, J., "Anisotropic Elasto-Plastic Finite Element Analysis of Thick and Thin Plates and Shells," International Journal for Numerical Methods in Engineering, Vol. 19, pp. 541-566, 1983.

111. Owen, D. and Figueiras, J., "Elasto-Plastic Analysis of Anisotropic Plates and Shells by the Semiloof Element," International Journal for Numerical Methods in Engineering, Vol. 19, pp. 521-539, 1983.

112. Marques, J. and Owen, D., "Strain Hardening Representation for Implicit Quasistatic Elasto-Viscoplastic Algorithms," Computers and Structures, Vol. 17, No. 2, pp. 301-304, 1983.

113. Lam, H, Choi, K. and Haug, E., "A Sparse Matrix Finite Element Technique for Iterative Structural Optimization," Computers and Structures, Vol. 16, No. 1-4, pp. 289-295, 1983.

114. Anand, S. and Shaw, R., "Mesh Refinement and Substructuring Technique in Elastic-Plastic Finite Element Analysis," Computers and Structures, Vol. 11, pp. 13-21, 1980.

115. Wissman, J. and Hauck, C., "Efficient Elastic-Plastic Finite Element Analysis with Higher Order Stress-Point Algorithms," Computers and Structures, Vol. 17, No. 1, pp. 89-95, 1983.

116. Bathe, K. and Bolourchi, S., "A Geometric and Material Nonlinear Plate and Shell Element," Computers and Structures, Vol. 11, pp. 23-48, 1980.

117. Thomas, T., Nair, S. and Garg, V., "A Numerical Study of Plasticity Models and Finite Element Types," Computers and Structures, Vol. 16, No. 5, pp. 669-675, 1983.

118. Carey, G. and Humphrey, D., "Mesh Refinement and Iterative Solution Methods for Finite Element Computations," International Journal for Numerical Methods in Engineering, Vol. 17, pp. 1717-1734, 1981.

119. Damjanio, F. and Owen, D., "Implicit Time Integration of Elasto-Viscoplastic Solids Subject to the Mohr-Coulomb Yield Criterion," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 1873-1881, 1982.

120. Nagtegall, J. and De Jong, J., "Some Computational Aspects of Elastic-Plastic Large Strain Analysis," International Journal for Numerical Methods in Engineering, Vol. 17, pp. 15-41, 1981.

121. Chaudonneret, M., "Structural Analysis in Viscoplasticity: Comparison Between Finite Element and Boundary Integral Equation Methods," Rech. Aerosp., 1980-6.

122. Haisler, W. and Stricklin, J., "Development and Evaluation of Solution Procedures for Geometrically Nonlinear Structural Analysis," AIAA Journal, March 1972.

123. Cook, R., "Loubignac's Iterative Method in Finite Element Elastostatics," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 67-75, 1982.

124. Dougalis, S. and Serbin, S., "On Some Unconditionally Stable, Higher Order Methods for the Numerical Solution of the Structural Dynamics Equations," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 1613-1621, 1982.

125. Kamat, M. and Vandenbrink, D., "A New Strategy for Stress Analysis Using the Finite Element Method," Computers and Structures, Vol. 16, No. 5, pp. 651-656, 1983.

126. Felippa, C. and Park, K., "Computational Aspects of Time Integration Procedures in Structural Dynamics," ASME, Journal of Applied Mechanics, Vol. 45, September 1978.

127. Park, K., "An Improved Semi-Implicit Method for Structural Dynamics Analysis," ASME, Journal of Applied Mechanics, Vol. 49, September 1982.

128. Akl, F., Dilger, W. and Irons, B., "Acceleration of Subspace Iteration," International Journal for Numerical Methods in Engineering, Vol. 18, pp.583-589, 1983.

129. Nour-Omid, B., Parlett, B. and Taylor, R., "Lanczos Versus Subspace Iteration for Solution of Eigenvalue Problems," International Journal for Numerical Methods in Engineering, Vol. 19, pp. 859-871, 1983.

130. Sanz-Serna, J., "On Finite Elements Simultaneously in Space and Time," International Journal for Numerical Methods in Engineering, Vol. 19, No.4, April 1983.

131. Day, M. and Yang, T., "A Mixed Variational Principle for Finite Element Analysis," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 1213-1230, 1982.

132. Krinke, D. and Houston, R., "An Analysis of Algorithms for Solving Differential Equations," Computers and Structures, Vol. 11, pp. 69-74, 1980.

133. Eisenstat, S., et al., "Yale Sparse Matrix Package I: The Symmetric Codes," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 1145-1151, 1982.

134. Peano, A., Szabo, B. and Mehta, A., "Self-Adaptive Finite Elements in Fracture Mechanics," Computer Methods in Applied Mechanics and Engineering, Vol. 16, pp. 69-80, 1978.

135. Felippa, C., "Solution of Linear Equations With Skyline-Stored Symmetric Matrix," Computers and Structures, Vol. 5, pp. 13-29, 1975.

136. Mondkar, D. and Powell, G., "Toward Optimal In-Core Equation Solving," Computers and Structures, Vol. 4, pp. 531-548, 1974.

137. Mondkar, D. and Powell, G., "Large Capacity Equation Solver for Structural Analysis," Computers and Structures, Vol. 4, pp. 699-728, 1974.

138. Mao-Hong, Y., "Twin Shear Stress Yield Criterion," Int. J. Mech., Sci., Vol. 25, No. 1, 1983.

139. Podgornyi, A., Gontarovskii, P. and Matyukhin, Y., "Creep Anisotropy in Finite Solids of Revolution," Strength of Materials, September 1981.

140. Kestin, J. and Rice, J., "Paradoxes in the Application of Thermodynamics to Strained Solids," Proceedings of the Critical Review of Thermodynamics Symposium, U. of Pittsburgh, 1969.

141. Eisenberg, M. and Yen, C., "A Theory of Multi-axial Anisotropic Viscoplasticity," ASME, Journal of Applied Mechanics, Vol. 48, June 1981.

142. Franklin, C., "A Model for Predicting Elevated Temperature-Time Dependent Material Behavior," I Mech E Conference Publications 1980-5, September 1980.

143. Martin, J., Topper, T. and Sinclair, G., "Computer Based Simulation of Cyclic Stress-Strain Behavior with Applications to Fatigue," Materials Research and Standards, February 1971.

144. Casey, J. and Naghdi, P., "On the Characterization of Strain-Hardening in Plasticity," ASME, Journal of Applied Mechanics, Vol. 48, June 1981.

145. Hashiguchi, K., "Constitutive Equations of Elasto-Plastic Materials with Anisotropic Hardening and Elastic-Plastic Transition," ASME, Journal of Applied Mechanics, Vol. 48, June 1981.

146. Bahei-El-Din, Y. and Dvorak, G., "Plasticity Analysis of Laminated Composite Plates," ASME, Journal of Applied Mechanics, Vol. 49, December 1982.

147. Shevchenko, Y., et. al., "Applicability of Certain Theories of Plasticity to Describing the Processes of Repeated Loading in Different Straight Trajectories," Strength of Materials, June 1980.

148. Levitas, V., "Some Models of Inelastic Deformation of Materials," Strength of Materials, December 1980.

149. Mohamed, F. and Langdon, T., "Deformation Mechanism Maps: Their Use in Predicting Creep Behavior," ASME, Journal of Engineering Materials and Technology, Paper 75-MAT-2, 1975.

150. Arshakuni, A., "Vector Variant of a Strain-Hardening Hypothesis and Kinetic Equations of High Temperature Creep," Strength of Materials, January 1981.

151. Lewis, D. and Hellen, T., "Analysis Techniques for Elevated Temperature Applications," I Mech E, Conference Publication 13 1973.

152. Arshakuni, A., "Selecting Equations of Transient Creep," Strength of Materials, September 1980.

153. Hseih, B., "On the Uniqueness and Stability of Endochronic Theory," ASME, Journal of Applied Mechanics, Vol. 47, December 1980.

154. Lee, D., "A Continuum Description of Anomalous Yielding in Anisotropic Materials," Metallurgical Transactions A, Vol. 9A, October 1978.

155. Lee, D. and Zaverl, F., "A Description of History Dependent Plastic Flow Behavior of Anisotropic Metals," ASME, Journal of Engineering Materials and Technology, Vol. 101, January 1979.

156. Shih, C. and Lee, D., "Further Developments in Anisotropic Plasticity," ASME, Journal of Engineering Materials and Technology, Vol. 100, July 1978.

157. Lee, D., "Anisotropic Yielding Behavior of a Fiber-Reinforced Directionally Solidified Eutectic Alloy," Metallurgical Transactions A, Vol. 9A, October 1978.

158. Lee, D. and Zaverl, F., "A Generalized Strain Rate Dependent Constitutive Equation for Anisotropic Metals," ACTA Metallurgica, Vol. 26, pp. 1771-1780, 1978.

159. Nikitenko, A., "Effect of Prior Creep Strain on Instantaneous Elasto-Plastic Deformation of a Material," Strength of Materials, February 1981.

160. Paslay, P. and Wells, C., "Uniaxial Creep Behavior of Metals Under Cyclic Temperature and Stress or Strain Variations," ASME, Journal of Applied Mechanics, September 1976.

161. Golub, V., "Effect of Cyclic and Static Stresses on the Cyclic Creep Process in Heat-Resistant Nickel Alloys Under Multicycle Load," Strength of Materials, March 1981.

162. Miller, A., "A Realistic Model for the Deformation Behavior of High-Temperature Materials," Fatigue at Elevated Temperatures, ASTM STP 520, American Society for Testing and Materials, 1973.

163. Milley, T. and Allen, D., "A Comparative Study of Nonlinear Rate-Dependent Mechanical Constitutive Equations for Crystalline Solids at Elevated Temperature," Master's Thesis, Virginia Polytechnic Institute and State University, VPI-E-82-5, 1982.

164. Bina, V. and Pech, R., "General Mathematical Model for the Creep Process," I Mech E Conference Publications 1980-5, September 1980.

165. Paslay, P., Wells, C. and Leverant, G., "An Analysis of Primary Creep of Nickel-Base Superalloy Single Crystals," ASME, Journal of Applied Mechanics, September 1970.

166. Wuttig, M. and Suzuki, T., "Time Dependent Stress-Strain Curves of Nonlinear Anelastic Solids," Metallurgical Transactions, Vol. 12A, Constitutive Modeling 3 I, July 1981.

167. Okrainets, P. and Pishchak, V., "Connection Between Structural Changes and Mechanical Behavior of Metal During Creep/Creep Equations," Strength of Materials, March 1980.

168. Gokheel, D.D., et. al, "Analysis of Progressive Accumulation of Unidirectional Strains With Thermal Cycling Under Creep Conditions," Strength of Materials, September 1980.

169. Shimazaki, Y. and Thompson, E., "Elasto Visco-Plastic Flow With Special Attention to Boundary Conditions," International Journal for Numerical Methods in Engineering, Vol. 17, pp. 97-112, 1981.

170. Pian, T., "Time-Independent Anisotropic Plastic Behavior by Mechanical Subelement Models," Paper Presented Symposium on Nonlinear Constitutive Equations for High Temperature Applications, Univ. of Akron, May 19-20, 1982.

171. Rees, D., "A Theory of Non-Linear Anisotropic Hardening," Proc. Instn. Mech. Engr., Vol. 1970, March 1983.

172. Nouailhas, D., Policella, H. and Kaczmarek, H., "On the Description of Cyclic Hardening Under Complex Loading Histories," Rech. Aerosp., 1982-6.

173. Robinson, D., "A Unified Creep-Plasticity Model for Structural Metals at High Temperature," ORNL/TM-5969, October 1978.

174. Robinson, D. and Swindeman, R., "Unified Creep-Plasticity Constitutive Equations for 2-1/4 CR-1 MO Steel at Elevated Temperature," ORNL/TM-8444, October 1982.

175. Lee, Y. and Male, A., "Plastic Deformation Analysis of Strain-Rate Sensitive Materials Under Plane Strain Conditions," Int. J. Mech. Sci., Vol. 25, No. 4, pp. 251-263, 1983.

176. Eisenberg, M. and Yen, C., "Application of a Theory of Viscoplasticity to Uniaxial Cyclic Loading," ASME, Journal of Engineering Materials and Technology, Vol. 105, April 1983.

177. Krempl, E., "Viscoplasticity Based on Total Strain. The Modelling of Creep with Special Considerations of Initial Strain and Aging," ASME, Journal of Engineering Materials and Technology, Vol. 101, October 1979.

178. Cassenti, B., "Material Response Predictions for Hot Section Gas Turbine Engine Components," Paper Presented AIAA/SAE/ASME 19th Joint Propulsion Conference, June 27-29, 1983.

179. Krieg, R., "A Practical Two Surface Plasticity Theory," ASME, Journal of Applied Mechanics, September 1975.

180. Miller, A., "An Inelastic Constitutive Model for Monotonic, Cyclic and Creep Deformation," ASME, Journal of Engineering Materials and Technology, April 1976.

181. Ostrom, F. and Lagneborg, R., "A Recovery-Athermal Glide Creep Model," ASME, Journal of Engineering Materials and Technology, April 1976.

182. Rohde, R. and Swearengen, J., "Deformation Modeling Applied to Stress Relaxation of Four Solder Alloys," ASME, Journal of Engineering Materials and Technology, Vol. 102, April 1980.

183. Kocks, U., "Laws for Work-Hardening and Low-Temperature Creep," ASME, Journal of Engineering Materials and Technology, January 1976.

184. Miller, A., "Modeling of Cyclic Plasticity with Unified Constitutive Equations: Improvements in Simulating Normal and Anomalous Bauschinger Effects," ASME, Journal of Engineering Materials and Technology, Vol. 102, April 1980.

185. Delph, T., "A Comparative Study of Two State-Variable Constitutive Theories," ASME, Journal of Engineering Materials and Technology, Vol. 102, October 1980.

186. Eisenberg, M., "A Generalization of Plastic Flow Theory with Application to Cyclic Hardening and Softening Phenomena," ASME, Journal of Engineering Materials and Technology, July 1976.

187. Lin, H. and Wu, H., "Strain-Rate Effect in the Endochronic Theory of Viscoplasticity," ASME, Journal of Applied Mechanics, March 1976.

188. Kennedy, C., Harms, W. and Douglas, D., "Multi-axial Creep Studies on Inconel at 1500° F," ASME, Journal of Basic Engineering, December 1959.

189. Henderson, J., "An Investigation of Multi-Axial Creep Characteristics of Metals," ASME, Journal of Engineering Materials and Technology, Vol. 101, October 1979.

190. Findley, W. Cho, U. and Ding, J., "Creep of Metals Under Combined Stresses, a Review," ASME, Journal of Engineering Materials and Technology, Vol. 101, October 1979.

191. Rohde, R. and Swearengen, J., "A Mechanical Equation of State for Inelastic Deformation of Iron: An Analytic Description," ASME, Journal of Engineering Materials and Technology, January 1977.

192. Swearengen, J. and Rohde, R., "Constitutive Modeling of Elevated Temperature Stress Relaxation in Austenitic Stainless Steels," ASME, Journal of Engineering Materials and Technology, Vol. 100, April 1978.

193. Hodge, P., "Structural Inelasticity XI - A Review of Some Piecewise Linear Theories of Plastic Strainhardening," Office of Naval Research Report AEM-H1-11, September 1974.

194. Hart, E., "Constitutive Relations for the Non-Elastic Deformation of Metals," General Electric Research and Development Report 75CRD168, August 1975.

195. Naghdi, P., "Some Constitutive Restrictions in Plasticity," Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects, AMD, Vol. 20, pp.79-93, ASME, 1976.

196. Ponter, A. and Walter, M., "A Theoretical and Experimental Investigation of Creep Problems with Variable Temperature," ASME, Journal of Applied Mechanics, December 1976.

197. Valanis, K., "Effect of Prior Deformation on Cyclic Response of Metals," ASME, Journal of Applied Mechanics, June 1974.

198. Liu, M., Krempl, E. and Nairn, D., "An Exponential Stress-Strain Law for Cyclic Plasticity," ASME, Journal of Engineering Materials and Technology, Paper No. 76-MAT-AA.

199. Krempl, E., Cernocky, E. and Liu, M., "The Representation of Viscoplastic Phenomena in Constitutive Equations," Constitutive Equations in Viscoplasticity, AMD, Vol. 20, ASME, 1976.

200. Leckie, F. and Hayhurst, D., "Constitutive Equations for Creep Rupture," ACTA Metallurgica, Vol. 25, pp. 1059-1070, 1977.

201. Daralias, Y. and Popov, E., "Plastic Internal State Variables of Cyclic Plasticity," ASME, Journal of Applied Mechanics, December 1976.

202. Yoshida, F., Murata, K and Shiratori, E., "Constitutive Equations of Cyclic Creep Under Increasing Stress Conditions," Bulletin of the JSME, Vol. 23, No. 177, March 1980.

203. Valanis, K. and Wu, H., "Endochronic Representation of Cyclic Creep and Relaxation of Metals," ASME, Journal of Applied Mechanics, March 1975.

204. Naghdi, P. and Trapp, J., "On the Nature of Normality of Plastic Strain Rate and Convexity of Yield Surfaces in Plasticity," ASME, Journal of Applied Mechanics, March 1975.

205. Takeuchi, S. and Argon, A., "Steady-State Creep of Alloys Due to Viscous Motion of Dislocations," ACTA Metallurgica, Vol. 24, pp. 883-889, 1976.

206. Stevens, R. and Flewitt, P., "The Dependence of Creep Rate on Microstructure in a Gamma Prime Strengthened Superalloy," ACTA Metallurgica, Vol. 29, pp. 867-882, 1981.

207. Okrainets, P. and Pishchak, V., "The Development of a Substructure and Internal Stresses During Creep," Phys. Met. Metall., Vol. 49, No. 1, pp. 129-134, 1980.

208. Yavari, P. and Langdon, T., "An Examination of the Breakdown in Creep by Viscous Glide in Solid Solution Alloys at High Stress Levels," ACTA Metallurgica, Vol. 30, pp. 2181-2196, 1982.

209. Mecking, H. and Kocks, U., "Kinematics of Flow and Strain-Hardening," ACTA Metallurgica, Vol. 29, pp. 1865-1875, 1981.

210. Raj, R. and Ghosh, A., "Micromechanical Modelling of Creep Using Distributed Parameters," ACTA Metallurgica, Vol. 29, pp. 283-292, 1981.

211. Nemat-Nasser, S. and Iwakuma, T., "Micromechanically Based Constitutive Relations for Polycrystalline Solids," Paper Presented at Symposium on Nonlinear Constitutive Equations for High Temperature Applications, Akron, Ohio, May 19-20, 1982.

212. Prinz, F., Argon, A. and Moffatt, W., "Recovery of Dislocation Structures in Plastically Deformed Copper and Nickel Single Crystals," ACTA Metallurgica, Vol. 30, pp. 821-830, 1982.

213. Reppich, B., "Some New Aspects Concerning Particle Hardening Mechanisms in Gamma Prime Precipitating Ni-Base Alloys -I. Theoretical Concept," ACTA Metallurgica, Vol. 30, pp. 87-94, 1982.

214. Yoshinaga, H., Horita, Z and Kurishita, H., "Determination of High-Temperature Deformation Mechanism in Crystalline Materials by the Strain-Rate Change Test," ACTA Metallurgica, Vol. 29, pp. 1815-1824, 1981.

215. Mohamed, F. and Langdon, T., "Deformation Mechanism Maps: Their Use in Predicting Creep Behavior," ASME, Journal of Engineering Materials and Technology, Paper No. 75-MAT-2.

216. Ashby, M., "A First Report on Deformation - Mechanism Maps," ACTA Metallurgica, Vol. 20, July 1972.

217. Leverant, G., Gell, M. and Hopkins, S., "The Effect of Strain Rate on the Flow Stress and Dislocation Behavior of a Precipitation-Hardened Nickel-Base Alloy," American Society for Metals, Materials Science and Engineering (8), 1971.

218. Sherby, O. and Miller, A., "Combining Phenomenology and Physics in Describing the High Temperature Mechanical Behavior of Crystalline Solids," ASME, Journal of Engineering Materials and Technology, Vol. 101, October 1979.

219. Spingarn, J. and Nix, D., "A Model for Creep Based on the Climb of Dislocations at Grain Boundaries," ACTA Metallurgica, Vol. 27, pp. 171-177, 1979.

220. Gibeling, J. and Nix, W., "Observations of Anelastic Backflow Following Stress Reductions During Creep of Pure Metals," ACTA Metallurgica, Vol. 29, pp. 1769-1784, 1981.

221. Purushothaman, S. and Tien, J., "Role of Back Stress in the Creep Behavior of Particle Strengthened Alloys," ACTA Metallurgica, Vol. 26, pp. 519-528, 1978, Constitutive Modelling and Microstructure 2 A.

222. Hill, R. and Rice, J., "Constitutive Analysis of Elastic-Plastic Crystals at Arbitrary Strain," J. Mech, Phys, Solids, Vol. 20, pp. 401-413, 1972.

223. Stouffer, D. and Abuelfoutouh, N., "A Potential Function Derivation of a Constitutive Equation for Inelastic Material Response," Paper Presented at Symposium on Nonlinear Constitutive Relations for High Temperature Applications, University of Akron, May 19-20, 1982.

224. Stouffer, D., "A Constitutive Representation for IN100," Air Force Materials Laboratory, AFWAL-TR-81-4039, January 1981.

225. Merzer, A. and Bodner, S., "Analytical Formulation of a Rate and Temperature Dependent Stress-Strain Relation," ASME, Journal of Engineering Materials and Technology, Vol. 101, July 1979.

226. Rosen, A. and Bodner, S., "The Influence of Strain Rate and Strain Aging on the Flow Stress of Commercially-Pure Aluminum," Journal of Mechanics and Physics of Solids, Vol. 15., pp. 47-62, 1967.

227. Anand, L., "Constitutive Equations for the Rate-Dependent Deformation of Metals at Elevated Temperatures," ASME, Journal of Engineering Materials and Technology, Vol. 104, January 1982.

228. Bodner, S. and Merzer A., "Viscoplastic Constitutive Equations for Copper with Strain Rate History and Temperature Effects," ASME, Journal of Engineering Materials and Technology, Vol. 100, October 1978.

229. Pattillo, P., Constitutive Equations for Rate Dependent Inelastic Behavior with Finite Deformations," PhD Dissertation, University of Notre Dame, University Microfilms, 1978.

230. Bassani, J., "Anisotropic Plasticity of Polycrystalline Metals," PhD Dissertation, Harvard University, May 1978.

231. Mondkar, D. and Powell, G., "Static and Dynamic Analysis of Nonlinear Structures," Earthquake Engineering Research Center, Report No. EERC 75-10, March 1975.

APPENDIX B

FORTRAN LISTING FOR MAIN CALLING PROGRAM

```
C ---------------------------------------------------
 SUBROUTINE MAINPG(LTOT)
C
C EQUATION SOLVER BY PACKED SKYLINE,COLUMN SOLUTION SCHEME
C STORAGE DYNAMIC ALLOCATION IS USED
C
CALCOM
C
 DIMENSION IA(1)
 COMMON A(3000)
 EQUIVALENCE (A,IA)
C
 EQUIVALENCE (W1,RPM123(1)),(W2,RPM123(2)),(W3,RPM123(3))
C
 CHARACTER JOBID*4(18)
 CHARACTER JUNK*4
C
 DIMENSION BUFF(400)
 CHARACTER IFILES*4(7)
 DATA IFILES(7)/';'/
C
C
 DATA MAXN/35/
C
 9000 FORMAT(V)
 9010 FORMAT(6A4)
 9100 FORMAT(1P6E12.3)
 9200 FORMAT(19A4)
 9300 FORMAT(//1X,70(1H#)//1X,18A4//1X,70(1H#)/)
 9400 FORMAT(A1)
C
 9510 FORMAT(/' NDIM  =',I4/'IANALY=',I4/' IPLANE=',I4/' INCOMI=',I4/
& ' TREF  =',F10.3/' ISOTHE=',I4/' ITHERM=',I4/
& ' NMAT  =',I4/' NMT   =',I4/' NMTCRP=',I4/'NORTHO=',I4/
& ' NSKEW =',I4/' NCONST=',I4//
& ' KFIXTY=',I4/' KPDIS =',I4/' KSKWBC=',I4/' KNTHK =',I4/
& ' KLOCAL=',I4/' KPRCNT=',I4//' MIDXYZ=',I4/
& ' KXYZCD=',I4/' KGAUSP=',I4/
& ' W1     =',F10.3/' W2     =',F10.3/' W3     =',F10.3/
& ' G1     =',F10.3/' G2     =',F10.3/' G3     =',F10.3/
& ' XBARAX=',F10.3/' XBARAY=',F10.3/' XBARAZ=',F10.3/)
 9520 FORMAT(/' NELTYP=',I4/' MAXEL =',I4/' NUMEL=',I4/
& ' MNODE =',I4/' MIDOFN=',I4/' MSTRAN=',I4/
& ' MAXNOD=',I4/' MNUMNP=',I4/' NUMNPS=',I4/' MAXEQ =',I8/)
C
 9530 FORMAT(/' KFIXTY=',I4/' NPDIS =',I4/' NSKWBC=',I4/
& ' KNTHK =',I4/' NLOCAL=',I4/' NPRCNT=',I4/)
C
 9540 FORMAT(/' NBLOCK=',I4/' NEQ    =',I8/' NEQBND=',I8/
& ' NTOTP1=',I8/)
C
 MTOT=LTOT
C
C SEQUENCIAL FILES IOVSTF AND NSTF
C      IOVSTF AND NSTF ARE FILE UNITS FOR OVER-ALL STIFFNESS
C         BEFORE AND AFTER BOUNDARY CONDITION MODIFICATION
C
C RANDOM-ACCESS FILE NRED
C
C      NRED IS FILE LOGIC UNIT FOR DECOMPOSED STIFFNESS MATRIX
C
 CALL CREATE(IFILE ,5000,0,IER)
 CALL CREATE(23,100,0,ISER)
 CALL CREATE(IOVSTF,5000,0,IER)
 CALL CREATE(NSTF  ,5000,0,IER)
 CALL CREATE(NRED  ,5000,1,IER)
 CALL CREATE(IFCONN,5000,1,IER)
```

```
 CALL CREATE(IFLK  ,5000,1,IER)
 CALL CREATE(IFSTIF,5000,1,IER)
 CALL CREATE(IFEF  ,5000,1,IER)
C
 PRINT,'INPUT DATA FILE'
 READ 9010,(IFILES(I),I=1,6)
 CALL ATTACH(IFILE,IFILES,3,0,ISTAT,BUFF)
C
 LAXDOF=10000
C
 METHOD=1
 EPSTOL=1.0E-5
 TOLEPS=0.01
 TOLDIS=2.5E-5
 BETAA =0.75
 BETAB =0.25
C
 READ(IFILE,9000,ERR=1) LINE
 REWIND IFILE
 READ(IFILE,9200) JUNK,JOBID
 LINEBS=1
 GO TO 2
 1 CONTINUE
 REWIND IFILE
 READ(IFILE,9200) JOBID
 LINEBS=0
 2 CONTINUE
 LINETP=LINEBS+1
 LINTP1=LINEBS+2
 LINTP2=LINEBS+3
 LINTP3=LINEBS+4
 LINTP4=LINEBS+5
 LINTP5=LINEBS+6
 LINTP6=LINEBS+7
 LINTP7=LINEBS+8
 LINTP8=LINEBS+9
 LINTP9=LINEBS+10
 PRINT 9300,JOBID
C READ(IFILE,9000) LINE,NDIM,IANALY,IPLANE,INCOMI,
C& TREF,ISOTHE,ITHERM,NMAT,NMT,NPS,NMTCRP,NORTHO,NSKEW,NCONST,
C& KFIXTY,KPDIS,KSKWBC,KNTHK,KLOCAL,KPRCNT,MIDXYZ,KXYZCD,KGAUSP,
C& ,W1,W2,W3,G1,G2,G3,XBARA
C
 CALL READZR(MAXN,NWORDS,A,IERR)
C
 NDIM  =A(LINEBS+1)
 IANALY=A(LINEBS+2)
 IPLANE=A(LINEBS+3)
 INCOMI=A(LINEBS+4)
 TREF  =A(LINEBS+5)
 ISOTHE=A(LINEBS+6)
 ITHERM=A(LINEBS+7)
C
 NMAT  =A(LINEBS+8)
 NMT   =A(LINEBS+9)
 NPS   =A(LINEBS+10)
 NMTCRP=A(LINEBS+11)
 NORTHO=A(LINEBS+12)
 NSKEW =A(LINEBS+13)
 NCONST=A(LINEBS+14)
C
 KFIXTY=A(LINEBS+15)
 KPDIS =A(LINEBS+16)
 KSKWBC=A(LINEBS+17)
 KNTHK =A(LINEBS+18)
 KLOCAL=A(LINEBS+19)
 KPRCNT=A(LINEBS+20)
```

```
 MIDXYZ=A(LINEBS+21)
 KXYZCD=A(LINEBS+22)
 KGAUSP=A(LINEBS+23)
C
 W1     =A(LINEBS+24)
 W2     =A(LINEBS+25)
 W3     =A(LINEBS+26)
 G1     =A(LINEBS+27)
 G2     =A(LINEBS+28)
 G3     =A(LINEBS+29)
 XBARA (1)=A(LINEBS+30)
 XBARA (2)=A(LINEBS+31)
 XBARA (3)=A(LINEBS+32)
C
 NGAUSS=2
 NGAUST=0
 CALL GAUSSO(NGAUSS,GAUSS,GAUSSW)
 IF(NDIM.EQ.2) GO TO 4
 NGAUST=NGAUSS
 CALL GAUSSO(NGAUST,GAUST,GAUSTW)
C
 4 CONTINUE
C
 IF(KGAUSP.EQ.0) GO TO 5
 MGAUSP=NGAUSS
 MGAUTP=NGAUST
 NPLOCA=MGAUSP**2*MAXO(1,MGAUTP)
 GO TO 6
 5 CONTINUE
 MGAUSP=1
 MGAUTP=1
 NPLOCA=1
 6 CONTINUE
C
 PRINT 9510,NDIM,IANALY,IPLANE,INCOMI,TREF,ISOTHE,ITHERM,
& NMAT,NMT,NMTCRP,NORTHO,NSKEW,NCONST,
& KFIXTY,KPDIS,KSKWBC,KNTHK,KLOCAL,KPRCNT,MIDXYZ,KXYZCD,
& KGAUSP,W1,W2,W3,G1,G2,G3,XBARA
C
 CALL COUNT
C
 PRINT 9520,NELTYP,MAXEL,NUMEL,MNODE,MIDOFN,MSTRAN,
& MAXNOD,MNUMNP,NUMNPS,MAXEQ
C
 REWIND IFILE
C
 READ(IFILE,9400) JUNK
 READ(IFILE,9400) JUNK
C
 CALL RANSIZ(IFCONN,MNODE,1)
 CALL RANSIZ(IFLK   ,MIDOFE,1)
 CALL RANSIZ(IFEF   ,MIDOFE,1)
 CALL RANSIZ(IFSTIF,MNES   ,1)
C
 IF(NDIM.LE.2) NGAUST=0
C
 IF(NCONST.EQ.0 .OR. KFCONS.LE.0) GO TO 10
 CALL CREATE(IFNCON,5000,1,IER)
 CALL CREATE(IFFCON,5000,1,IER)
 CALL CREATE(IFECON,5000,1,IER)
C
 CALL RANSIZ(IFNCON,LCONLK,1)
 CALL RANSIZ(IFFCON,LCONLK,1)
 CALL RANSIZ(IFECON,LCONES,1)
C
 GO TO 15
 10 CONTINUE
```

```
IFNCON=-IFNCON
IFFCON=-IFFCON
IFECON=-IFECON
15 CONTINUE
C
NDIMP1=NDIM+1
NDIMSQ=NDIM*NDIM
C
W1SQ=W1**2
W2SQ=W2**2
W3SQ=W3**2
WMATRX(1,1)=W2SQ+W3SQ
WMATRX(2,2)=W1SQ+W3SQ
WMATRX(3,3)=W1SQ+W2SQ
WMATRX(1,2)=-W1*W2
WMATRX(1,3)=-W1*W3
WMATRX(2,3)=-W2*W3
WMATRX(2,1)=WMATRX(1,2)
WMATRX(3,1)=WMATRX(1,3)
WMATRX(3,2)=WMATRX(2,3)
C
GMATRX(1)=G1
GMATRX(2)=G2
GMATRX(3)=G3
C
CALL ADRESS
C
PRINT 9530,KFIXTY,NPDIS,NSKWBC,KNTHK,NLOCAL,NPRCNT
C
PRINT 9540,NBLOCK,NEQ,NEQBND,NTOTP1
C
IF(IANALY.EQ.1) GO TO 60
C
CALL CREATE(IFSS  ,5000,1,IER)
CALL RANSIZ(IFSS  ,LRSS3D,1)
C
C      INITIALIZATION FOR IN-ELASTIC IFSS DATA
C
       TOTCRT=0.
DO 40 I=1,LRSS3D
A(I)=0.0
40 CONTINUE
       A(112)=TREF
C
IPRECD=0
DO 50 IEL=1,NUMEL
DO 50 I=1,NPLOCA
IPRECD=IPRECD+1
CALL GENIOF(2,IFSS,IPRECD,LRSS3D,LRSS3D,A,A(IPSS))
50 CONTINUE
C
60 CONTINUE
C
LCASE=0
IFIRST=0
C
C      INITIALIZE TOTAL MECHANICAL LOADS AT TIME (T)
C                 TOTAL DISPLACEMENTS AT TIME (T)
C                 INCREMENTAL LOAD FOR THE CURRENT INCREMENT TIME
C
DO 100 I=1,NEQ
A(IPFM1B+I)=0.0
A(IPDISB+I)=0.0
A(IPDFB +I)=0.0
100 CONTINUE
C
C      INITIALIZE NODAL TEMPERATURES AND TEMPERATURE GRADIENTS
```

```
C
 IPOUTB=IPOUTN-1
 DO 150 I=1,NUMNPS
 IA(IPOUTB+I)=0
 A(IPTN1B+I)=TREF
 A(IPTN2B+I)=TREF
 IF(KNTHK.EQ.0) GO TO 150
 A(IPDT1B+I)=0.0
 A(IPDT2B+I)=0.0
 150 CONTINUE
C
C     INITIALIZE ELEMENT MATERIAL #
C
 DO 160 I=1,NUMEL
 IEL=I
 CALL FLAG(2,1,IDMATN,1,IA(IPFLGB+IEL))
 160 CONTINUE
C
C     TO INITIALIZE ELEMENT STIFFNESS RE-COMPUTATION CODE
C                       ELEMENT AVERAGE TEMPERATURE
C
 ISTIFF=1
C
 IPTEMB=IPTEME-1
C
 DO 170 JEL=1,NUMEL
 IEL=JEL
 CALL FLAG(2,1,IDSTIF,1,IA(IPFLGB+IEL))
 A(IPTEMB+IEL)=TREF
 170 CONTINUE
C
 200 CONTINUE
C
 LCASE=LCASE+1
C
       PRINT,"LOAD CASE",LCASE
 DO 300 I=1,NEQ
 A(IPFM2B+I)=0.0
 300 CONTINUE
C
C     TO INPUT LOAD PARAMETERS AND LOADS
C
 CALL LOADS (IA(IPFLAG),IA(IPICON),IA(IPJCON),IA(IPEINT),IA(IPNINT),
& A(IPDOFB),A(IPLM  ),A(IPFM1 ),A(IPTN1 ),A(IPDTN1),
&                  A(IPFM2 ),A(IPTN2 ),A(IPDTN2),TOTCRT,DELT,IER)
C
       PRINT,"DONE OF LOADS"
 IF(IER.NE.0) GO TO 990
C
C PRINT,'FM1',(A(I),I=IPFM1,(IPFM1+NEQM1))
C PRINT,'FM2',(A(I),I=IPFM2,(IPFM2+NEQM1))
C
 DO 900 INC=1,LODINC
C
       TOTCRT=TOTCRT+DELT
C PRINT,'INC,ISTIFF,NTEMCH',INC,ISTIFF,NTEMCH
C
C     CURRENT TOTAL MECHANICAL LOADS
C
 DO 360 I=1,NEQ
 A(IPFOB+I)=A(IPFM1B+I)+A(IPFM2B+I)
 360 CONTINUE
C PRINT,'FO',(A(I),I=IPFO,(IPFO+NEQM1))
C PRINT,'NTEMCH',NTEMCH
C
 IF(NTEMCH.EQ.0) GO TO 400
C
```

```
C      CURRENT NODAL TEMPERATURES AND THE GRADIENTS
C
 DO 370 I=1,NUMNPS
 A(IPTNB+I)=A(IPTN1B+I)+A(IPTN2B+I)
 IF(KNTHK.EQ.0) GO TO 370
 A(IPDTNB+I)=A(IPDT1B+I)+A(IPDT2B+I)
 370 CONTINUE
C
C PRINT,'TN',(A(I),I=IPTN,(IPTN+NUMNPS-1))
C
C      TO COMPUTE ELEMENT AVERAGE TEMPERATURE
C
 CALL ELTEMP(IA(IPFLAG),IA(IPICON),IA(IPJCON),IA(IPNINT),
&          A(IPTN),A(IPTEME))
C
C PRINT,'TEME',(A(I),I=IPTEME,(IPTEME+NUMEL-1))
C
 400 CONTINUE
C
 IF(ISTIFF.EQ.0) GO TO 500
C
 CALL ELSTIF(A(IPFLAG))
C PRINT,'DONE OF ELSTIF'
C
C TO ASSEMBLE OVER-ALL STIFFNESS AND DETERMINE MAX. DIAGONAL VALUE
C
C
C      CALL ADDSTF(NEQBND,A(IPFLAG),LK,ES,MAXMIN,MAXA,NCOLBV,AARRAY)
 CALL ADDSTF(NEQBND,A(IPFLAG),A(IPDOFE),A(IPSTIF),A(IPMXMN),A(IPMAXA),
&  A(IPCLBV),A(IPMATA))
C PRINT,'DONE OF ADDSTF'
C
C TO MODIFY OVER-ALL STIFFNESS DUE TO SPECIFIE DISP. B.C.
C
C      CALL MODSTF(NEQBND,MAXA,NEQDIS,NCOLBV,AARRAY)
 CALL MODSTF(NEQBND,A(IPMATA))
C PRINT,'DONE OF MODSTF'
C
C      TO DECOMPOSE THE MODIFIED OVER-ALL STIFFNESS
C
 CALL DECOMP
C
 500 CONTINUE
C
C PRINT,'FO',(A(I),I=IPFO,(IPFO+NEQM1))
C PRINT,'FM1',(A(I),I=IPFM1,(IPFM1+NEQM1))
C      TO COMPUTE INCREMENTAL PSEUDO THERMO-MECHANICAL LOADS
C
C      A(IPDF) IS A WORKING ARRAY
C
 CALL INCLOD(A(IPFO),A(IPFM1),A(IPDF),A(IPDFO))
C
C PRINT,'INCREMENTAL ELASTIC THERMO-MECHANICAL LOADS'
C PRINT,(A(I),I=IPDFO,(IPDFO+NEQM1))
C
 IF(IANALY.NE.1) GO TO 600
C
C      ELASTIC ANALYSIS
C
 CALL ELASTY
 GO TO 800
C
C      IN-ELASTIC ANALYSIS
C
 600 CONTINUE
C
 CALL PLASTY(MSTRAN,NDIM,NPS,NMT,IA(IPICON),IA(IPJCON),
```

```
& IA(IPDOFE),IA(IPNINT),A(IPDF),A(IPDFO),A(IPFP),A(IPDFP),
& A(IPDFP1),A(IPDIS1),A(IPDFC),A(IPTN),A(IPDTN),A(IPSKEW),
& A(IPTSKW),A(IPXYZ),A(IPTHKN),A(IPFLAG),A(IPELVO),A(IPTEME),
& A(IPTMAT),A(IPORTH),A(IPSXSX),DELT)
C
 800 CONTINUE
C
 CALL UPDATE
C
 IF(KDISP .LT.O .AND. KSTRES.LT.O) GO TO 850
 CALL OUTPUT(INC,LODINC)
C
 850 CONTINUE
C
 IFIRST=1
C
 IF(NTEMCH.NE.O) GO TO 900
C
 ISTIFF=O
C
 900 CONTINUE
C
C      INITIALIZATIONS FOR THE NEXT LOAD CASE
C
 ISTIFF=O
 DO 910 JEL=1,NUMEL
 IEL=JEL
 CALL FLAG(2,1,IDSTIF,O,IA(IPFLGB+IEL))
 910 CONTINUE
C
 GO TO 200
C
 990 CONTINUE
 STOP
 END
```

APPENDIX C

LISTING OF INPUT DATA

Following is a detailed description of the input for the 8 noded shell programs and 20 noded brick programs. Input for the 9 noded shell program is similar to that for the 8 noded shell.

```
C------------------------------------------------------------------
C        INPUT FOR MSS8 WITH H-A PLASTICITY AND SIMPLE CREEP
C------------------------------------------------------------------
C SUBROUTINE MAINPG
C
C    NOTE  THE FOLLOWING INPUTS ARE ALL ON ONE LINE FOLLOWING THE TITLE
C      IANALY:  analysis flag
C               .lt.0 read additional line containing restart info
C               =1 (in absolute value) elastic
C               =2 (in absolute value) plastic
C               =3 (in absolute value) creep
C               =23 (in absolute value) creep and plasticity combined
C      TREF:   reference temperature
C      ISOTHE: isothermal element flag
C               =0 isothermal
C               .ne.0 shape functions used to interpolate int.pt. temperatures
C      ITHERM: thermal load flag
C               =0 no thermal loads
C               .ne.0 thermal loads calculated
C      NMAT:    number of materials
C      NMT:     number of temperatures for material property specification
C      NPS:     number of points used to specify stress-strain curves
C      NMTCRP:  number of temperatures where creep coefficients specified
C      NORTHO:  orthotropic material flag
C               =0 isotropic material
C               .ne.0 orthotropic material (mat'l props. in 3 directions req'd)
C                Program stores number of different ortho mat'ls in NORTHO
C      NSKEW:   skewed coordinate systems flag
C               =0 no skewed coordinate systems
C                ne 0 skewed coordinate systems
C                Program stores number of skewed coord. systems in NSKEW.
C      MASSCD:   mass matrix flag
C                =0 no mass matrix calculated
C                =1 lumped mass matrix calculated
C                =2 consistent mass matrix calculated
C     KFIXITY:   fixed node flag
C                =0 no fixed nodes
C                .ne.0 there are some fixed nodes
C                 System of eqns is reduced by number of fixed DOF
C      KPDIS.   prescribed nodal displacement flag
C      KSKWBC:  skewed boundary condition flag
C               =0 no skewed BC
C               .ne.0 skewed BC's
C      KLOCAL:  local coordinate system flag
C               =0 no local coord. systems
C               .ne.0 there are local coord. systems (5 max)
C      KPRCNT:  nodal percentage distribution flag
C               =0 no percentage distributions
C               .ne.0 there are percentage distributions of nodes (5 sets max)
C      MIDXYZ:  flag to generate element mid side nodes
C               =0 no mid side node generation
C               .ne.0 mid side node generation
C      KXYZCD:  nonstandard coordinate input flag
C               =0 standard coordinates used
C               .ne.0 nonstandard coordinate order is used
C               Standard is (X,Y,Z) for cartesian
C                           (R,THETA,Z) for cylindrical
C                           (R,THETA,PHI) for spherical
C      KGAUSP:  gauss integration order flag
C               =0 first order integration
C               .ne.0 second order integration
C      W1,W2,W3: unit vector defining axis of rotation in global system
C      G1,G2,G3: unit vector defining gravity direction in global system
C      XBARAX,XBARAY,XBARAZ: location of origin of rotational coord.
C                  system in global system
C      KEIGTP:   eigenvalue/eigenvector computation flag
```

```
C               =0 and MASSCD ne 0 calculate eigenvalue/eigenvector
C                                  using determinant search method
C               =1 and MASSCD ne 0 calculate eigenvalue eigenvector
C                                  using subspace iteration method
C----------------------------------------------------------------------
C   IF(IANALY LT 0)
C      NOUT,LASCAS,NEXCAS
C
C      NOUT:    output (for later restart) file creation flag
C            =0 no output file created
C            =1 output file for restart created
C      LASCAS  load case on restart file from which solution proceeds
C      NEXCAS  load case on input file which is next load case to solve
C----------------------------------------------------------------------
C SUBROUTINE ADRESS
C**********************************************************************
C  SUBROUTINE EKIND2
C
C      IEL1,IEL2,IEL3,. ......
C      0
C
C      IELn:     element numbers
C                If element entered as negative, all elements from previous
C                entered element to the negative element are included
C......................................................................
C  SUBROUTINE NODOF2
C
C      IDOFN,NODE1,NODE2,NODE3, ....
C      0
C
C      IDOFN  DOF per node
C      NODEn: nodes with this number of DOF
C......................................................................
C  SUBROUTINE CONNEC
C**********************************************************************
C   SUBROUTINE ELGEN2
C
C      IEL,(INODE(I),I=1,NODE),IEEND1,IEINC1,ININC1,IEEND2,IEINC2,ININC2
C      0
C
       IEL:    beginning element number
C      INODE:  array sized to number of nodes per element and is
C              connectivity of element in terms of global node numbers
C      IEEND1: ending element number for generation set 1
C      IEINC1: increment in element number for generation set 1
C      ININC1: increment in node numbers for generation set 1
C      IEEND2: ending element number for generation set 2
C      IEINC2: increment in element number for generation set 2
C      ININC2: increment in node numbers for generation set 2
C
C......................................................................
C  SUBROUTINE XYZCOR
C
C   IF(KLOCAL.NE.0)
C     XYZO(1,NLOCAL),XYZO(2,NLOCAL),XYZO(3,NLOCAL)
C     0
C
C     XYZO: array 3 by number of local coordinate systems and is
C           origin of local coordinate system in global system (5 max)
C
C   IF(KPRCNT.NE.0)
C     NPTS,(PERCNT(I,NPRCNT),I=1,NPTS)
C     0
```

```
      NPTS    number of nodes generated in this set
      PERCNT  arra, NPTS x number of percent distribution sets and is
              distribution of nodes between coordinates input below
    IF(KXYZCD.NE.0)
      ICDXYZ(1),ICDXYZ(2),ICDXYZ(3)

      ICDXYZ: array specifying coordinate input order
    IF(INC1.EQ.0)
      NODENO,X1,X2,X3,INC1,INC2,ISYSNO,ISYSTP,IPCNT1,IPCNT2
      O

      NODENO: beginning node number
      X1,X2,X3: coordinates
      INC1:   node number increment for generation set 1
      INC2:   node number increment for generation set 2
      ISYSNO: local coordinate system number used for above coordinates
      ISYSTP: global coordinate system type
              =0 cartesian
              =1 cylindrical
              =2 spherical
      IPCNT1: percentage distribution set for node generation set 1
      IPCNT2: percentage distribution set for node generation set 2
    IF(INC1.NE.0)
      NODENO,X1,X2,X3

      NODENO: ending node number for node generation set 1
      X1,X2,X3: ending coordinates
    IF(INC2.NE.0)
      NODENO,X1,X2,X3

      NODENO: ending node number for node generation set 2
      X1,X2,X3: ending coordinates
***********************************************************************

    IF(MIDYYZ.NE.0)
    SUBROUTINE NODMID

      IEL1,IEL2,IEL3.........
      O

      IELn:   elements for mid-side node generation
***********************************************************************

    SUBROUTINE NODEIN

      THICK,NO1,NO2,NO3.................
      O

      THICK:  thickness
      NOn:    nodes with this thickness
***********************************************************************

    SUBROUTINE MTABLE

      SPECWT,ISOCOD,TMAT(1,NMAT),TMAT(2,NMAT),.....,TMAT(NMT,NMAT)
      O

      SPECWT:  specific weight = density * g  (LB/CUBIC INCH)
      ISOCOD:  isotropic material code
```

```
                   =0 isotropic
                    ne 0 orthotropic
        TMAT    array number of mat 1 temps by number of mat 's and 's
                temperatures where material properties are specified

      IF(ISOCOD.EQ.0) INPUT NMT OF THE FOLLOWING LINES
        E1,NU1,ALPHA

        E1:     Young s modulus (program multiplies by 1E6)
        NU1:    Poisson's ratio
        ALPHA:  thermal coefficient (program multiplies by 1E-6)

      IF(ISOCOD.NE.0) INPUT NMT OF THE FOLLOWING LINES
        E1,E2,E3,NU1,NU2,NU3,ALPHA

        En:   Young's modulus in the n direction (multiplies by 1E6)
        NUn: Poisson's ratio in the n direction
        ALPHA: thermal coefficient (multiplies by 1E-6)

      IF(NPS.NE.0)  INPUT NMT OF THE FOLLOWING LINES
        STSXSX(1,1,NMT,NMAT),STSXSX(2,1,NMT,NMAT)
        STSXSX(1,2,NMT,NMAT),STSXSX(2,2,NMT,NMAT)
                     .                      .
                     .                      .
                     .                      .
        STSXSX(1,NPS,NMT,NMAT),STSXSX(2,NPS,NMT,NMAT)
        BETA(1,NMAT),BETA(2,NMAT),. . .BETA(NMT,NMAT)

        STSXSX: array 2 by NPS by NMT by NMAT and is
                stress,strain pairs defining curve at temp  NMT
        BETA   array NMT by NMAT and is
               hardening coefficient at temperature NMT
               =1 isotropic hardening only
               =0 kinematic hardening only
               (0 .LE. BETA .LE. 1)

      IF(NMTCRP.NE.0) INPUT NMT OF THE FOLLOWING LINES
        TEMP,SNORM,Q,R,STRCUT

        TEMP:   temperature at which these properties apply
        SNORM:  normalizing stress for these properties
        Q:      creep property
        R:      creep property  (creep = Q * (stress) ** R)
        STRCUT: cutoff stress below which no creep occurs

**********************************************************************

      IF(NSKEW.NE.0)
     SUBROUTINE SKEW

        ISKEWN,A1,A2,A3
        0

        ISKEWN:  skew set number
                 .gt.0  A's are node numbers defining skewed coord. system
                 .lt.0  A's are successive rotation angles in degrees
        A1,A2,A3: nodes or angles defining skewed coordinate system
                  if nodes, axis1 is formed from A1 to A2
                            axis2 is formed from A1 to A3
                            axis3 is formed as axis1 cross axis3

**********************************************************************

      IF(KSKWBC.NE.0)
     SUBROUTINE SKEWBC

        ISKEWN,NODE1,NODE2,NODE3,. . . . . . . . . .
```

```
C      C
C
C      ISKEWN.   skew set number
C      NODEn:    nodes where skew BC's referred to this skew set are applied
C
C*******************************************************************************
C
C    IF(NORTHO NE.O)
C   SUBROUTINE ORTHOP
C
C      ISKEWN.IEL1.IEL2.IEL3.....  ....
C      C
C
C      ISKEWN.   skew set number
C      IELn.     elements whose material properties have this skew set orienta
C
C*******************************************************************************
C
C    IF(KFIXITY.NE.O)
C   SUBROUTINE FIXITY
C
C      IDIREC.NODE1.NODE2.NODE3......   . . .
C      O
C
C      IDIREC:   global fixed direction
C      NODEn:    nodes with this fixity
C
C*******************************************************************************
C
C    IF(KPDIS.NE.O)
C   SUBROUTINE PREDIS
C
C      IDIREC.VALUE.NODE1.NODE2.NODE3.   . ..   ....
C      O
C
C      IDIREC:   global direction of prescribed displacement
C      VALUE:    magnitude of prescribed displacement
C      NODEn:    nodes with this displacement
C
C-------------------------------------------------------------------------------
C
C SUBROUTINE LOADS
C
C      KDISP.KSTRES.LODINC.MATCH.NTEMCH.NREMOV.NCLOAD.RPM.
C            KGLOAD.KTLOAD.KALOAD.KLLOAD.KCLOAD.
C            METHOD.EPSTOL.TOLEPS.TOLDIS
C
C      KDISP:   nodal data print flag
C               =1 print nodal data (displacements)
C                  as specified in this load case
C               =0 print nodal data as specified previously
C               =-1 no nodal data printing
C      KSTRES:  element data print flag
C               =1 print element data (stresses.strains)
C                  as specified in this load case
C               =0 print element data as specified previously
C               =-1 no element data printing
C      LODINC:  number of load increments
C               .lt.0 sets ITIME=1 (reads time data)
C      MATCH:   material change flag
C               =1 there is material changed
C               =0 no material is changed
C      NTEMCH:  termperature change flag
C               =1 there is change in temperature
C               =0 no temperature change
C      NREMOV:  element removal flag
C               =1 some elements are to be removed
```

```
C      ALPHA:  thermal coefficient (program multiplies by 1E-6)
C
C    IF(ISOCOD.NE.0) INPUT NMT OF THESE LINES
C      E1,E2,E3,NU1,NU2,NU3,ALPHA
C
C      En:    Young's modulus in the n direction (multiplies by 1E6)
C      NUn:   Poisson's ratio in the n direction
C      ALPHA: thermal coefficient (multiplies by 1E-6)
C
C    IF(NPS.NE.0) INPUT NMT OF THESE LINES
C      STSXSX(1,1,NMT,NMAT),STSXSX(2,1,NMT,NMAT)
C      STSXSX(1,2,NMT,NMAT),STSXSX(2,2,NMT,NMAT)
C
C
C
C      STSXSX(1,NPS,NMT,NMAT),STSXSX(2,NPS,NMT,NMAT)
C
C      STSXSX:  array 2 by NPS by NMT by NMAT and is
C               stress,strain pairs defining curve at temp. NMT
C
C    IF(NMTCRP.NE.0) INPUT NMT OF THESE LINES
C      TEMP,SNORM,Q,R,STRCUT
C
C      TEMP:   temperature at which these properties apply
C      SNORM:  normalizing stress for these properties
C      Q:      creep property
C      R:      creep property  (creep = Q * (stress) ** R)
C      STRCUT: cutoff stress below which no creep occurs
C*******************************************************************
C
C    IF(NSKEW.NE.0)
C   SUBROUTINE SKEW
C
C      ISKEWN,A1,A2,A3
C      0
C
C      ISKEWN:    skew set number
C                 gt.0  A's are node numbers defining skewed coord. system
C                 lt.0  A's are successive rotation angles in degrees
C      A1,A2,A3:  nodes or angles defining skewed coordinate system
C                 if nodes, axis1 is formed from A1 to A2
C                           axis2 is formed from A1 to A3
C                           axis3 is formed as axis1 cross axis3
C*******************************************************************
C
C    IF(KSKWBC.NE.0)
C   SUBROUTINE SKEWBC
C
C      ISKEWN,NODE1,NODE2,NODE3,......
C      0
C
C      ISKEWN:  skew set number
C      NODEn:   nodes where skew BC's referred to this skew set are applied
C
C*******************************************************************
C
C    IF(NORTHO.NE.0)
C   SUBROUTINE ORTHOP
C
C      ISKEWN,IEL1,IEL2,IEL3,........
C      0
C
C      ISKEWN:  skew set number
C      IELn:    elements whose material properties have this skew set orienta
C
```

```
C*************************************************************************
C
C   IF(KFIXITY NE 0)
C  SUBROUTINE FIXITY
C
C     IDIREC,NODE1,NODE2,NODE3,
C     0
C
C     IDIREC    global fixed direction
C     NODEn     nodes with this fixity
C
C*************************************************************************
C
C   IF(KPDIS NE 0)
C  SUBROUTINE PREDIS
C
C     IDIREC,VALUE,NODE1,NODE2,NODE3,
C     0
C
C     IDIREC    global direction of prescribed displacement
C     VALUE     magnitude of prescribed displacement
C     NODEn     nodes with this displacement
C
C-------------------------------------------------------------------------
C
C SUBROUTINE LOADS
C
C     KDISP,KSTRES,LODINC,MATCH,NTEMCH,NREMOV,NCLOAD,RPM,TEND,
C          KGLOAD,KTLOAD,KALOAD,KLLOAD,KCLOAD,
C          METHOD,EPSTOL,TOLEPS,TOLDIS
C
C     KDISP     nodal data print flag
C               =1 print nodal data (displacements)
C                  as specified in this load case
C               =0 print nodal data as specified previously,
C               =-1 no nodal data printing
C     KSTRES    element data print flag
C               =1 print element data (stresses,strains)
C                  as specified in this load case
C               =0 print element data as specified previously,
C               =-1 no element data printing
C     LODINC:   number of load increments
C     MATCH     material change flag
C               =1 there is material changed
C               =0 no material is changed
C     NTEMCH:   temperature change flag
C               =1 there is change in temperature
C               =0 no temperature change
C     NREMOV:   element removal flag
C               =1 some elements are to be removed
C               =0 no elements removed
C     NCLOAD:   concentrated load flag
C               =1 there are concentrated loads
C               =0 no concentrated loads
C     RPM:      revolutions per minute for this load case
C     TEND:     time at end of load step
C               used only when time dependent response (creep) desired
C     KGLOAD:   gravity load flag
C               =1 there are gravity loads
C               =0 no gravity loads
C     KTLOAD    thermal load flag
C               =1 thermal loads calculated
C               =0 no thermal loads
C     KALOAD:   area load flag
C               =1 there are area loads
C               =0 no area loads
C     KLLOAD:   line load flag
```

```
C                =1 there are line loads
C                =0 no line loads
C       TOLEPS.  convergence tolerance on strain
C       TOLDIS.  convergence tolerance on displacement
C
C***********************************************************************
C
C    IF(KDISP.EQ.1)
C   SUBROUTINE OUTPUTN
C
C      IDISP,IREF,NO1,NO2,NO3, . . .
C      O
C
C      IDISP   print flag
C              =0 don t print
C              =1 do print
C      IREF
C      NOn     nodes for which output is desired
C
C***********************************************************************
C
C    IF(KSTRES.EQ 1)
C   SUBROUTINE OUTPUTN
C
C      IELF,ISTRESS,ISTRAIN,LOCA,IEL1,IEL2,IEL3,
C      O
C
C      IELF:      force print flag
C                 =1 print element forces
C                 =0 don t print forces
C      ISTRESS:   stress print flag
C                 =1 print stresses
C                 =0 don t print stresses
C      ISTRAIN:   strain print flag
C                 =1 print strains
C                 =0 don t print strains
C      LOCA     location for which data is printed
C               =1 center of gravit. of element
C               =2 element nodes
C               =3 Gauss points
C      IELn     elements for which output desired
C
C***********************************************************************
C
C    IF(MATCH.NE.O)
C   SUBROUTINE INPUT2
C
C      MATNO,IEL1,IEL2,IEL3, . . . . . . . . .
C      O
C
C      MATNO:  new material number
C      IELn:   elements changed to this material
C
C***********************************************************************
C
C    IF(NTEMCH.NE.O)
C   SUBROUTINE NODEIN
C
C      TEMP,NO1,NO2,NO3, . . . . .
C      O
C
C      TEMP:  new temperature
C      NOn:   nodes changed to this temperature
C
C
C***********************************************************************
C
```

```
C    IF(NTEMCH.NE.0)
C   SUBROUTINE NODEIN
C
C      DTEMP,NO1,NO2,NO3,
C      0
C
C      DTEMP   temperature gradient
C      NOn     nodes with this temperature gradient
C
C******************************************************************
C
C    IF(NREMOV.NE.0)
C   SUBROUTINE INPUT1
C
C      IEL1,IEL2,IEL3,
C      0
C
C      IELn  removed elements
C
C******************************************************************
C
C    IF(NCLOAD.NE.0)
C   SUBROUTINE CLOADS
C
C      IDIREC,VALUE,NO1,NO2,NO3,
C      0
C
C      IDIREC:  global direction of load
C      VALUE:   magnitude of load
C      NOn      nodes where this concentrated load applied
C
C******************************************************************
C
C    IF(KELOAD.NE.0)
C   SUBROUTINE ELOADS
C
C==================================================================
C
C    IF(KALOAD.NE.0)
C   SUBROUTINE ALOADS
C
C      IEBEG,IEEND,IEINC,IFACE,IDIRE,P1,P2,P3,P4
C      0
C
C      IEBEG:  beginning element
C      IEEND:  ending element
C      IEINC:  increment in element numbers
C      IFACE:  element face where pressure load applied
C      IDIRE:  direction of load in element local coord. system
C      P1,P2,P3,P4: corner pressure values in clockwise? direction
C
C##################################################################
C
C    IF(KLLOAD.NE.0)
C   SUBROUTINE LLOADS
C
C      IEBEG,IEEND,IEINC,IFACE,IDIR,P1,P2
C      0
C
C      IEBEG:  beginning element
C      IEEND:  ending element
C      IEINC:  increment in element numbers
C      IFACE:  element face where line load applied
C      IDIR:   direction of load in element local coord. system
C      P1,P2:  pressures at either end of line load
C
C##################################################################
```

```
C*****************************************************************
C-----------------------------------------------------------------
C   IF(KPDIS NE 0 AND LCASE NE 1)
C SUBROUTINE PREDIS
C
C     IDIREC,VALUE,NODE1,NODE2,NODE3,..
C     0
C
C     IDIREC.  global direction of prescribed displacement
C     VALUE    magnitude of prescribed displacement
C     NODEn.   nodes with this displacement
C
C      NOTE.  THE NODES WITH PRESCRIBED DISPLACEMENTS MUST BE THE
C             SAME THROUGHOUT AND MUST BE SPECIFIED FROM THE CUTSET
C
C-----------------------------------------------------------------
```

```
C------------------------------------------------------------------
C        INPUT FOR MSS8 WITH BODNER S CONSTITUTIVE MODEL
C------------------------------------------------------------------
C SUBROUTINE MAINPG
C
C     NOTE  THE FOLLOWING ARE ALL ON ONE LINE AFTER THE TITLE LINE
C
C       IANAL:  analysis flag
C                lt 0 read additional line containing restart info
C               =1 (in absolute value) elastic
C               =2 (in absolute value) inelastic
C       TREF    reference temperature
C       ISOTHE  isothermal element flag
C               =0 isothermal
C                ne.0 shape functions used to interpolate int pt  temperatures
C       ITHERM  thermal load flag
C               =0 no thermal loads
C                ne 0 thermal loads calculated
C       NMAT    number of materials
C       NMT     number of temperatures for material property specification
C       NMTCRP  number of temperatures where creep coefficients specified
C       NORTHO  orthotropic material flag
C               =0 isotropic material
C                ne.0 orthotropic material (mat 1 props. in 3 directions req d)
C                Program stores number of different ortho mat'ls in NORTHO
C       NSKEW   skewed coordinate systems flag
C               =0 no skewed coordinate systems
C                ne 0 skewed coordinate systems
C                Program stores number of skewed coord  systems in NSKEW
C       MASSCD   mass matrix flag
C               =0 no mass matrix will be computed
C               =1 lumped mass matrix computed
C               =2 consistent mass matrix computed
C       KFIXITY  fixed node flag
C               =0 no fixed nodes
C                ne.0 there are some fixed nodes
C                System of eqns is reduced by number of fixed DOF
C       KPDIS   prescribed nodal displacement flag
C       KSKWBC  skewed boundary condition flag
C              =0 no skewed BC
C               ne 0 skewed BC s
C       KLOCAL  local coordinate system flag
C               =0 no local coord. systems
C                ne.0 there are local coord  systems (5 max)
C       KPRCNT  nodal percentage distribution flag
C               =0 no percentage distributions
C                ne.0 there are percentage distributions of nodes (5 sets max)
C       MIDXYZ: flag to generate element mid side nodes
C               =0 no mid side node generation
C                ne 0 mid side node generation
C       KXYZCD: nonstandard coordinate input flag
C               =0 standard coordinates used
C                ne.0 nonstandard coordinate order is used
C               Standard is (X,Y,Z) for cartesian
C                           (R,THETA,Z) for cylindrical
C                           (R,THETA,PHI) for spherical
C       KGAUSP: gauss integration order flag
C               =0 first order integration
C               .ne.0 second order integration
C       W1,W2,W3: unit vector defining axis of rotation in global system
C       G1,G2,G3: unit vector defining gravity direction in global system
C       XBARAX,XBARAY,XBARAZ: location of origin of rotational coord.
C                 system in global system
C       KEIGTP:  eigenvalue/eigenvector computation flag
C               =0.and.MASSCD.ne.0 calculate eigenvalue/eigenvector
C                                  using determinant search method
C               =1.and.MASSCD.ne.0 calculate eigenvalue/eigenvector
```

```
C                                       using subspace iteration method
C
C--------------------------------------------------------------------------
C SUBROUTINE ADRESS
C*************************************************************************
C  SUBROUTINE EKIND2
C
C      IEL1,IEL2,IEL3,..
C      0
C
C      IELn:     element numbers
C                If element entered as negative, all elements from previous
C                entered element to the negative element are included
C
C*************************************************************************
C  SUBROUTINE NODOF2
C
C      IDOFN,NODE1,NODE2,NODE3,
C      0
C
C      IDOFN:  DOF per node
C      NODEn:  nodes with this number of DOF
C
C*************************************************************************
C  SUBROUTINE CONNEC
C*************************************************************************
C   SUBROUTINE ELGEN2
C
C      IEL,(INODE(I),I=1,NODE),IEEND1,IEINC1,ININC1,IEEND2,IEINC2,ININC2
C      0
C
C      IEL:    beginning element number
C      INODE   array sized to number of nodes per element and is
C              connectivity of element in terms of global node numbers
C      IEEND1:  ending element number for generation set 1
C      IEINC1:  increment in element number for generation set 1
C      ININC1:  increment in node numbers for generation set 1
C      IEEND2:  ending element number for generation set 2
C      IEINC2   increment in element number for generation set 2
C      ININC2:  increment in node numbers for generation set 2
C
C*************************************************************************
C  SUBROUTINE XYZCOR
C
C    IF(KLOCAL.NE.0)
C      XYZO(1,NLOCAL),XYZO(2,NLOCAL),XYZO(3,NLOCAL)
C      0
C
C      XYZO: array 3 by number of local coordinate systems and is
C            origin of local coordinate system in global system (5 max)
C
C    IF(KPRCNT.NE.0)
C      NPTS,(PERCNT(I,NPRCNT),I=1,NPTS)
C      0
C
C      NPTS:   number of nodes generated in this set
C      PERCNT: array NPTS x number of percent distribution sets and is
C              distribution of nodes between coordinates input below
C
C    IF(KXYZCD.NE.0)
C      ICDXYZ(1),ICDXYZ(2),ICDXYZ(3)
C
C      ICDXYZ: array specifying coordinate input order
C
C    IF(INC1.EQ.0)
C      NODENO,X1,X2,X3,INC1,INC2,ISYSNO,ISYSTP,IPCNT1,IPCNT2
C      0
```

```
C
C      NODENO  beginning node number
C      X1,X2,X3  coordinates
C      INC1    node number increment for generation set 1
C      INC2    node number increment for generation set 2
C      ISYSNO  local coordinate system number used for above coordinates
C      ISYSTP  global coordinate system type
C              =0 cartesian
C              =1 cylindrical
C              =2 spherical
C      IPCNT1  percentage distribution set for node generation set 1
C      IPCNT2  percentage distribution set for node generation set 2
C
C    IF(INC1.NE.0)
C      NODENO,X1,X2,X3
C
C      NODENO  ending node number for node generation set 1
C      X1,X2,X3  ending coordinates
C
C    IF(INC2.NE.0)
C      NODENO,X1,X2,X3
C
C      NODENO  ending node number for node generation set 2
C      X1,X2,X3  ending coordinates
C
C**********************************************************************
C
C   IF(MIDXYZ.NE.0)
C   SUBROUTINE NODMID
C
C      IEL1,IEL2,IEL3,
C      0
C
C      IELn    elements for mid-side node generation
C
C**********************************************************************
C
C   SUBROUTINE NODEIN
C
C      THICK,NO1,NO2,NO3,  ....  ....
C      0
C
C      THICK   thickness
C      NOn     nodes with this thickness
C
C**********************************************************************
C
C   SUBROUTINE MTABLE
C
C      SPECWT,ISOCOD,TMAT(1,NMAT),TMAT(2,NMAT),  ....,TMAT(NMT,NMAT)
C      0
C
C      SPECWT:  specific weight = density * g
C      ISOCOD:  isotropic material code
C               =0 isotropic
C               .ne.0 orthotropic
C      TMAT:    array number of mat'l temps by number of mat'ls and is
C               temperatures where material properties are specified
C
C    IF(ISOCOD.EQ.0) INPUT NMT OF THESE LINES
C      E1,NU1,ALPHA
C
C      E1:     Young's modulus (program multiplies by 1E6)
C      NU1:    Poisson's ratio
C      ALPHA:  thermal coefficient (program multiplies by 1E-6)
C
C    IF(ISOCOD.NE.0) INPUT NMT OF THESE LINES
```

```
C      E1,E2,E3,NU1,NU2,NU3,ALPHA
C
C      En     Young s modulus in the n direction (multiplies by 1E6)
C     *Nun    Poisson s ratio in the n direction
C      ALPHA  thermal coefficient (multiplies by 1E-6)
C
C
C   IF(NMTCRP.NE.0) INPUT NMT OF THESE LINES
C
C      TEMP  D  AN  Z0  Z1  Z2  AM  A  R
C
C      TEMP = temperature in degrees F
C      D, AN, Z0, Z1, Z2, AM, A, R = material parameters required
C              for Bodner's model at temperature TEMP
C              Z0,Z1,Z2 in KSI,  AM in 1/KSI,  remaining are dimensionless
C
C*****************************************************************************
C
C   IF(NSKEW.NE.0)
C  SUBROUTINE SKEW
C
C      ISKEWN,A1,A2,A3
C      0
C
C      ISKEWN    skew set number
C                gt.0  A s are node numbers defining skewed coord. system
C                lt.0  A s are successive rotation angles in degrees
C      A1,A2,A3  nodes or angles defining skewed coordinate system
C                if nodes, axis1 is formed from A1 to A2
C                          axis2 is formed from A1 to A3
C                          axis3 is formed as axis1 cross axis3
C
C*****************************************************************************
C
C   IF(KSKWBC.NE.0)
C  SUBROUTINE SKEWBC
C
C      ISKEWN,NODE1,NODE2,NODE3,
C      0
C
C      ISKEWN    skew set number
C      NODEn     nodes where skew BC s referred to this skew set are applied
C*****************************************************************************
C
C   IF(NORTHO.NE.0)
C  SUBROUTINE ORTHOP
C
C      ISKEWN,IEL1,IEL2,IEL3,.....
C      0
C
C      ISKEWN:   skew set number
C      IELn:     elements whose material properties have this skew set orienta
C
C*****************************************************************************
C
C   IF(KFIXITY.NE.0)
C  SUBROUTINE FIXITY
C
C      IDIREC,NODE1,NODE2,NODE3, .......
C      0
C
C      IDIREC:   global fixed direction
C      NODEn:    nodes with this fixity
C
C*****************************************************************************
```

```
C
C   IF(KPDIS NE 0)
C  SUBROUTINE PREDIS
C
C      IDIREC,VALUE,NODE1,NODE2,NODE3,
C      O
C
C      IDIREC    global direction of prescribed displacement
C      VALUE     magnitude of prescribed displacement
C      NODEn     nodes with this displacement
C
C-----------------------------------------------------------------
C
C SUBROUTINE LOADS
C
C      KDISP,KSTRES,LODINC,MATCH,NTEMCH,NREMOV,NCLOAD,RPM,
C           KGLOAD,KTLOAD,KALOAD,KLLOAD,KCLOAD,
C           METHOD,EPSTOL,TOLEPS,TOLDIS
C
C      KDISP    nodal data print flag
C               =1 print nodal data (displacements)
C                  as specified in this load case
C               =0 print nodal data as specified previously
C               =-1 no nodal data printing
C      KSTRES   element data print flag
C               =1 print element data (stresses,strains)
C                  as specified in this load case
C               =0 print element data as specified previously
C               =-1 no element data printing
C      LODINC   number of load increments
C                1t 0 sets ITIME=1 (reads time data)
C      MATCH    material change flag
C               =1 there is material changed
C               =0 no material is changed
C      NTEMCH   termperature change flag
C               =1 there is change in temperature
C               =0 no temperature change
C      NREMOV   element removal flag
C               =1 some elements are to be removed
C               =0 no elements removed
C      NCLOAD:  concentrated load flag
C               =1 there are concentrated loads
C               =0 no concentrated loads
C      RPM      revolutions per minute for this load case
C      KGLOAD   gravity load flag
C               =1 there are gravity loads
C               =0 no gravity loads
C      KTLOAD   thermal load flag
C               =1 thermal loads calculated
C               =0 no thermal loads
C      KALOAD:  area load flag
C               =1 there are area loads
C               =0 no area loads
C      KLLOAD:  line load flag
C               =1 there are line loads
C               =0 no line loads
C      TOLEPS:  convergence tolerance on strain
C      TOLDIS:  convergence tolerance on displacement
C
C****************************************************************
C
C   IF(KDISP.EQ.1)
C  SUBROUTINE OUTPUTN
C
C      IDISP,IREF,NO1,NO2,NO3,................
C      O
C
```

```
C      IDISP: print flag
C             =0 don't print
C             =1 do print
C      IREF
C      NOn    nodes for which output is desired
C
C*******************************************************************
C
C   IF(KSTRES EQ 1)
C  SUBROUTINE OUTPUTN
C
C     IELF,ISTRESS,ISTRAIN,LOCA,IEL1,IEL2,IEL3,
C     0
C
C     IELF:      force print flag
C                =1 print element forces
C                =0 don't print forces
C     ISTRESS:   stress print flag
C                =1 print stresses
C                =0 don't print stresses
C     ISTRAIN:   strain print flag
C                =1 print strains
C                =0 don't print strains
C     LOCA      location for which data is printed
C               =1 center of gravity of element
C               =2 element nodes
C               =3 Gauss points
C     IELn:     elements for which output desired
C
C*******************************************************************
C
C   IF(ITIME NE 0)
C  SUBROUTINE DYNTIM
C
C     N2M,TCRP,TINIT,ECMAX,SIGMAX,ERMAX,DELTMIN,DELTMULT
C
C     N2M      number of equal time steps in this load case
C              =0 dynamic time incrementing is used
C     TCRP:    total time in this load case
C  following inputs apply for dynamic time incrementing only
C     TINIT:   initial time step
C              =0 and initial load case, TINIT=DELTMIN
C              =0 and not initial load case, TINIT= 5 * last time step of
C                 previous load case
C     ECMAX:   maximum inelastic strain increment desired in any time step
C              default=.0001
C     SIGMAX:  maximum change in stress desired in any time step
C              default=1000 psi
C     ERMAX:   maximum estimated integration error allowed in any time step
C              default=.01 (1%)
C     DELTMIN: minimum allowable time step
C                default=.001 * TCRP
C     DELTMULT: maximum multiple of current time step allowed for next time
C                default=1.5
C
C*******************************************************************
C
C   IF(MATCH.NE.0)
C  SUBROUTINE INPUT2
C
C     MATNO,IEL1,IEL2,IEL3,.......  .....
C     0
C
C     MATNO: new material number
C     IELn:  elements changed to this material
C
C*******************************************************************
```

```
C
C    IF(NTEMCH NE 0)
C   SUBROUTINE NODEIN
C
C      TEMP,NO1,NO2,NO3,
C      0
C
C      TEMP   new temperature
C      NOn    nodes changed to this temperature
C
C**********************************************************************
C
C    IF(NTEMCH NE 0)
C   SUBROUTINE NODEIN
C
C      DTEMP,NO1,NO2,NO3,
C      0
C
C      DTEMP   temperature gradient
C      NOn:    nodes with this temperature gradient
C
C**********************************************************************
C
C    IF(NREMOV NE 0)
C   SUBROUTINE INPUT1
C
C      IEL1,IEL2,IEL3,
C      0
C
C      IELn:  removed elements
C
C**********************************************************************
C
C    IF(NCLOAD NE 0)
C   SUBROUTINE CLOADS
C
C      IDIREC,VALUE,NO1,NO2,NO3,
C      0
C
C      IDIREC:  global direction of load
C      VALUE:   magnitude of load
C      NOn:     nodes where this concentrated load applied
C
C**********************************************************************
C
C    IF(KELOAD NE 0)
C   SUBROUTINE ELOADS
C
C######################################################################
C
C    IF(KALOAD NE 0)
C    SUBROUTINE ALOADS
C
C      IEBEG,IEEND,IEINC,IFACE,IDIRE,P1,P2,P3,P4
C      0
C
C      IEBEG:  beginning element
C      IEEND:  ending element
C      IEINC:  increment in element numbers
C      IFACE:  element face where pressure load applied
C      IDIRE:  direction of load in element local coord. system
C      P1,P2,P3,P4:  corner pressure values in clockwise? direction
C
C######################################################################
C
C    IF(KLLOAD NE 0)
```

```
C     SUBROUTINE LLOADS
C
C       IEBEG,IEEND,IEINC,IFACE,IDIR,P1,P2
C       C
C
C       IEBEG   beginning element
C       IEEND   ending element
C       IEINC   increment in element numbers
C       IFACE:  element face where line load applied
C       IDIR:   direction of load in element local coord  system
C       P1,P2:  pressures at either end of line load
C
C*********************************************************************
C.....................................................................
C---------------------------------------------------------------------
C     IF(KPDIS.NE.0 AND LCASE.NE.1)
C    SUBROUTINE PRECIS
C
C       IDIREC,VALUE,NODE1,NODE2,NODE3,.
C       C
C
C       IDIREC.   global direction of prescribed displacement
C       VALUE.    magnitude of prescribed displacement
C       NODEn.    nodes with this displacement
C
C        NOTE:   THE NODES WITH PRESCRIBED DISPLACEMENTS MUST BE THE
C                SAME THROUGHOUT AND MUST BE SPECIFIED FROM THE OUTSET
C
C--  ---------------------------------------------------------------
```

```
1000C              ORGANIZATION OF INPUT
1010C        FOR T3CYAN WITH H-A PLASTICITY AND SIMPLE CREEP
1020C
1030C  I HEADING AND CONTROL INFORMATION
1040C    I.1 TITLE CARD
1050C    I.2 PROBLEM SIZING
1060C    I.3 ANALYSIS AND RESTART OPTIONS
1070C    I.4 EQUATION NUMBERING AND BANDING OPTIONS
1080C
1090C  II NODE COORDINATES AND TRANSFORMATIONS
1100C    II.1 NODE COORDINATES
1110C    II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
1120C
1130C  III ELEMENT DEFINITION
1140C    III.1 HEADER LINE FOR ELEMENT
1150C    III.2 20 NODED SOLID DEFINITION
1160C
1170C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
1180C    IV.1 LOAD CASE CONTROL CARD
1190C    IV.2 ACCELERATION SPECIFICATIONS
1200C
1210C  V MATERIAL PHYSICAL PROPERTIES
1220C    V.1 MATERIAL PHYSICAL PROPERTIES
1230C      V.1.1 ISOTROPIC ELASTIC PROPERTIES
1240C      V.1.2 ORTHOTROPIC ELASTIC PROPERTIES
1250C    V.2 ORTHOTROPIC AXES ORIENTATION TABLE
1260C    V.3 INELASTIC MATERIAL CHARACTERIZATION
1270C
1280C  VI TIME AND TIME INCREMENTING CONTROL
1290C
1300C  VII CONVERGENCE CRITERIA
1310C
1320C  VIII INITIAL NODAL CONSTRAINED DISPLACEMENTS
1330C
1340C  IX INITIAL NODAL APPLIED FORCES
1350C
1360C  X INITIAL NODAL TEMPERATURES
1370C
1380C  XI INITIAL ELEMENT PRESSURE LOADS
1390C
1400C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
1410C    XII.1 LOAD CASE CONTROL CARD
1420C    XII.2 ACCELERATION SPECIFICATION
1430C
1440C  XIII FINAL NODAL CONSTRAINED DISPLACEMENTS
1450C
1460C  XIV FINAL NODAL APPLIED FORCES
1470C
1480C  XV FINAL NODAL TEMPERATURES
1490C
1500C  XVI FINAL ELEMENT PRESSURE LOADS
1510C
1520C
1530C------------------------------------------------------------------
1540C  I HEADING AND CONTROL INFORMATION
1550C
1560C    I.1 TITLE CARD
1570C
1580C LINE  ITITLE
1590C
1600C   ITITLE = ANY 1 TO 72 CHARACTER TITLE FOR THE ANALYSIS
1610C
1620C******************************************************************
1630C    I.2 PROBLEM SIZING
1640C
1650C LINE  NUMNP  NM  IT  RTEM  NLC
1660C
```

```
C    NUMNP  : NUMBER OF STRUCTURAL NODES ( ENTER AS A NEGATIVE NUMBER FOR
C              TIMING SUMMARY )
C    NM     : NUMBER OF DIFFERENT MATERIALS ( MAXIMUM=3 )
C    IT     : THERMAL STRESS OPTION
C             = 0  INCLUDE THERMAL LOADS
C             = 1  IGNORE THERMAL LOADS
C    RTEM   : REFERENCE TEMPERATURE ( DEGREES F )
C    NLC    : NUMBER OF LOAD CASES
C
C*********************************************************************
C     I.3 ANALYSIS AND RESTART OPTIONS
C
C LINE  LAWCRP  NOUT  NRESTA  INREST  MASSCD
C
C    LAWCRP : TYPE OF INELASTIC ANALYSIS
C              = 0  ELASTIC ANALYSIS
C              = 1  HAISLER-ALLEN PLASTICITY
C              = 2  SECONDARY CREEP MODEL
C              =12  PLASTICITY AND CREEP COMBINED
C    NOUT    : OUTPUT FILE CREATION OPTION
C              = 0  DO NOT CREATE OUTPUT FILE
C              = 1  CREATE OUTPUT FILE
C    NRESTA  : RESTART OPTION
C              = 0  THIS IS NOT A RESTART RUN
C              > 0  INPUT THE LOAD CASE FROM WHICH THE RESTART IS TO PROCEED
C    NOTE:(OUTPUT FROM THIS CASE MUST HAVE BEEN PREVIOUSLY PUT ON AN OUTPUT
C      FILE.  THE FIRST NEW LOAD CASE WILL BE LABELED AS NRESTA + 1.)
C
C    INREST  : LOAD CASE NUMBER IN THE CURRENT INPUT FILE WHICH BECOMES
C             THE FIRST NEW LOAD CASE TO BE SOLVED WHEN RESTARTING, WHERE
C             ( 1.LE.INREST.LE.NLC ). IF 0 IS INPUT INREST = 1 IS ASSUMED
C
C    MASSCD  : MASS MATRIX FLAG
C              = 0  DO NOT CREATE MASS MATRIX
C              = 1  CREATE LUMPED MASS MATRIX
C              = 2  CREATE CONSISTENT MASS MATRIX
C
C*********************************************************************
C     I.4 EQUATION NUMBERING AND BANDING OPTIONS
C
C LINE  N  IBAND  IPBAND
C
C    N       : KEY CODE
C             = 0 NO NUMBERING OR BANDING
C             = -1 ACTIVATE NUMBERING AND BANDING OPTION
C    IBAND   : BANDING OPTION
C             = 0 USE DEFAULT OPTION
C            = 1 ASSUME NODE NUMBER IS THE SAME AS MATRIX POSITION
C            = 2 ASSUME INPUT NODE ORDER DEFINES MATRIX POSITION
C    IPBAND  : PRINTOUT OPTION
C            = 0 NO PRINTOUT OF EQUATION NUMBERS
C            = 1 PRINT OUT THE EQUATION NUMBER FOR EACH DEGREE OF FREEDOM
C
C
C---------------------------------------------------------------------
C  II NODE COORDINATES AND TRANSFORMATIONS
C
C*********************************************************************
C    II.1 NODE COORDINATES
C
C     (ENTER THE FOLLOWING LINE FOR EACH NODE)
C
C LINE  N  X  Y  Z
C
C   N : NODE NUMBER
C   X : NODE X COORDINATE
C   Y : NODE Y COORDINATE
```

```
C   Z : NODE Z COORDINATE
C
C  (AFTER THE LAST NODE ENTER THE FOLLOWING LINE)
C
C LINE  O  NSKEW
C   NSKEW : INDICATOR FOR LOCAL NODE COORDINATE SYSTEM TRANSFORMATION
C           = O NONE WILL BE INPUT
C           = 1 ONE OR MORE WILL BE INPUT
C
C*****************************************************************
C    II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
C
C  IF(NSKEW.GT.O)
C     ( ENTER ALL NODES WITH LOCAL COORDINATE SYSTEM TRANSFORMATIONS)
C
C LINE  N  NO  NI  NJ
C LINE O
C
C   N  : NODE NUMBER HAVING A ROTATED LOCAL COORDINATE SYSTEM
C   NO : NODE ON LOCAL X AXIS
C   NI : NODE ON LOCAL X AXIS IN THE +X DIRECTION FROM NO
C   NJ : NODE IN LOCAL XY PLANE SUCH THAT THE +Z AXIS IS IN THE
C        DIRECTION OF THE VECTOR P3 = (NO-NI) X (NO-NJ). THE +Y AXIS IS
C        IN THE DIRECTION OF THE VECTOR P2 = P3 X (NO-NI)
C
C
C
C-----------------------------------------------------------------
C  III ELEMENT DEFINITION
C
C*****************************************************************
C    III.1 HEADER LINE FOR ELEMENT DATA
C
C LINE  IPRINT
C   IPRINT : ELEMENT DATA EXTENDED PRINTOUT OPTION
C            = O NONE
C            = 1 DUMP VOLUMES AND DISTORTION PARAMETERS
C            = 2 ALSO DUMP ELEMENT STIFFNESSES
C            = 3 ALSO DUMP ELEMENT EQUIVALENT NODAL FORCES AND MASSES
C
C*****************************************************************
C    III.2 20 NODED SOLID DEFINITION
C
C     (ENTER THE FOLLOWING LINES FOR EACH ELEMENT)
C
C LINE  NEL  N1  N2  N3  N4  N5  N6  N7  N8  N9  N10  N11  N12
C LINE  N13  N14  N15  N16  N17  N18  N19  N20  IMAT  IOR
C LINE NO  NP  NQ    ( OPTIONAL LINE, ENTER ONLY IF IOR = 100)
C
C LINE  O
C
C   NEL  : ELEMENT NUMBER
C   N1 ... N20 : NODES DEFINING THE ELEMENT  ( SEE FIG III.2 )
C   IMAT : MATERIAL NUMBER
C   IOR  : ORTHOTROPIC MATERIAL AXIS SYSTEM INDICATOR
C         = O  ISOTROPIC MATERIAL
C         (1.LE.IOR.LE.99) IOR IS ORIENTATION IDENTIFIER OF SYSTEM
C                          GIVEN UNDER MATERIAL DATA SECTION V.2
C         = 100 ORTHOTROPIC AXES DEFINED BY OPTIONAL LINE OF INPUT
C
C
C
C-----------------------------------------------------------------
C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
C
C*****************************************************************
C    IV.1 LOAD CASE CONTROL CARD
```

```
C
C LINE  RPM  IAXIS  IACC  NPCI  NTI  ITHST
C
C   RPM    : ROTATIONAL SPEED IN RPM
C   IAXIS  : GLOBAL AXIS ABOUT WHICH STRUCTURE IS ROTATING
C            = 1 X AXIS
C            = 2 Y AXIS
C            = 3 Z AXIS
C   IACC   : INDICATOR FOR INPUT OF ACCELERATON LOADS IN GROUP IV.2
C            = 0 NO INPUT
C            = 1 ACCELERATION LOADS ARE INPUT
C   NPCI : INDICATOR FOR A CHANGE IN MATERIAL PROPERTIES
C            = 0 NO CHANGE TO MATERIAL PROPERTIES
C            = 1 CHANGE ELASTIC PROPERTIES ( GROUPS V.1 AND V.2)
C            = 2 CHANGE INELASTIC PROPERTIES ( GROUP V.3 )
C            = 12 CHANGE ELASTIC AND INELASTIC PROPERTIES
C   NOTE: IF THIS IS FIRST LOAD CASE, PROGRAM SETS NPCI=12
C
C   NTI    : NUMBER OF NODAL TEMPERATURES WHICH ARE RESPECIFIED.
C            = 0 NO RESPECIFIED TEMPERATURES, STIFFNESS IS RECOMPUTED
C            = -1 NO RESPECIFIED TEMPERATURES, STIFFNESS IS NOT RECOMPUTED
C            > 0 NTI RESPECIFIED TEMPERATURES, STIFFNESS IS RECOMPUTED
C   ITHST  : TOTAL STRAIN PRINTOUT OPTION
C            = 0 INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C            = 1 DO NOT INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C
C*******************************************************************
C    IV.2 ACCELERATION SPECIFICATION FOR INERTIAL OR GRAVITY LOADS
C
C    IF(IACC.GT.0)
C LINE  ACCELX  ACCELY  ACCELZ
C
C   ACCELX : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL X
C   ACCELY : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Y
C   ACCELZ : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Z
C
C
C-------------------------------------------------------------------
C  V MATERIAL PHYSICAL PROPERTIES
C
C*******************************************************************
C    V.1 ELASTIC CONSTANTS
C
C   IF(NPCI.EQ.1.OR.NPCI.EQ.12)
C     (ENTER NM OF THE FOLLOWING LINES)
C
C LINE  MTN  NMT  DEN
C
C   MTN : MATERIAL NUMBER ( MTN.GE.1.AND.MTN.LE.NM )
C        (IF MATERIAL IS ISOTROPIC, INPUT MTN AS A NEGATIVE NUMBER
C         TO SIMPLIFY INPUT)
C   NMT : NUMBER OF TEMPERATURES AT WHICH ELASTIC PROPERTIES WILL BE
C         GIVEN FOR THIS MATERIAL
C   DEN : WEIGHT DENSITY OF THE MATERIAL ( POUNDS/IN**3 )
C
C###################################################################
C    V.1.1 ISOTROPIC MATERIAL
C
C   IF(MTN.LT.0)
C   ( ENTER NMT OF THESE LINES )
C
C LINE  TEMP  E  PR  AL
C
C   TEMP : TEMPERATURE ( DEGREES F. )
C   E    : ELASTIC MODULUS ( 10**6 P.S.I. )
C   PR   : POISSON'S RATIO
C   AL   : MEAN COEFFICIENT OF THERMAL EXPANSION ( 10**-6 IN/IN-DEG. F. )
```

```
C
C#####################################################################################
C     V.1.2 ORTHOTROPIC MATERIAL
C
C    IF(MTN.GT.O)
C     (ENTER NMT OF THE FOLLOWING LINES)
C     (DIRECTIONS 1,2,3 CORRESPOND TO THE MATERIAL ORTHOTROPIC AXES X',Y',Z')
C
C LINE TEMP E11 E22 E33 NU12 NU13 NU23 G12 G23 G31 AL1 AL2 AL3
C
C    TEMP : TEMPERATURE AT WHICH PROPERTIES ARE GIVEN ( DEG. F. )
C    E11  : ELASTIC MODULUS IN THE 1 DIRECTION
C    E22  : ELASTIC MODULUS IN THE 2 DIRECTION
C    E33  : ELASTIC MODULUS IN THE 3 DIRECTION
C    NU12 : POISSON'S RATIO RELATING DIRECTIONS 1 AND 2
C    NU13 : POISSON'S RATIO RELATING DIRECTIONS 1 AND 3
C    NU23 : POISSON'S RATIO RELATING DIRECTIONS 2 AND 3
C    G12  : SHEAR MODULUS IN THE 1-2 PLANE
C    G23  : SHEAR MODULUS IN THE 2-3 PLANE
C    G31  : SHEAR MODULUS IN THE 3-1 PLANE
C    AL1  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 1 DIRECTION
C    AL2  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 2 DIRECTION
C    AL3  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 3 DIRECTION
C
C*********************************************************************************
C     V.2 ORTHOTROPIC AXIS ORIENTATION TABLE
C
C LINE  NOR
C
C   NOR : NUMBER OF ORIENTATION SPECIFICATIONS
C         (O.LE.NOR.LE.10) i.e. MAX OF 10 ORIENTATION SYSTEMS ALLOWED
C     (NOTE: IF THE MATERIALS ARE ISOTROPIC OR THE ORTHOTROPIC AXES
C               COINCIDE WITH THE GLOBAL AXES, ENTER O)
C     ENTER O. ( MAXIMUM OF 10 SPECIFICATIONS ALLOWED )
C
C   IF(NOR.GT.O)
C  ( ENTER NOR LINES OF THE FOLLOWING )
C
C LINE  I  NO  NP  NQ
C
C   I             : ORIENTATION IDENTIFIER ( IOR IN ELEMENT INPUT )
C   NO, NP, NQ : NODE NUMBERS IDENTIFYING ORTHOTROPIC AXES ( SEC. II.2 )
C
C*********************************************************************************
C  V.3 INELASTIC MATERIAL PROPERTIES
C
C     (IF NPCI.EQ.2.OR.NPCI.EQ.12 )
C     (IF LAWCRP.EQ.1.OR.LAWCRP.EQ.12)
C     (ENTER MTN OF THE NEXT FOUR LINE SETS - MTN,SSTEMP,PPV,AND BET)
C
C LINE  MTN  NPTS  NTM
C
C      MTN  : MATERIAL NUMBER
C      NPTS : NUMBER OF STRESS-STRAIN POINTS PER CURVE
C      NTM  : NUMBER OF TEMPERATURES FOR WHICH STRESS-STRAIN CURVES ARE GIVEN
C
C
C LINE  SSTEMP(1)  SSTEMP(2) ..... SSTEMP(NTM)
C
C      SSTEMP : ARRAY OF TEMPERATURES WHERE STRESS-STRAIN CURVES ARE GIVEN
C                 (IN INCREASING ORDER)
C
C LINE  PPV(MTN,I,1,1)  PPV(MTN,I,1,2)
C LINE  PPV(MTN,I,2,1)  PPV(MTN,I,2,2)
C             .               .
C             .               .
C             .               .
```

```
C LINE  PPV(MTN,I,NPTS,1)  PPV(MTN,I,NPTS,2)
C
C     PPV(MTN,I,J,1) : STRESS VALUES IN INCREASING ORDER FROM J=1 TO NPTS
C                      FOR TEMPERATURE I AND MATERIAL MTN
C     PPV(MTN,I,J,2) : STRAIN VALUES IN INCREASING ORDER FROM J=1 TO NPTS
C                      FOR TEMPERATURE I AND MATERIAL MTN
C
C LINE  BET(1)  BET(2) ..... BET(NTM)
C
C     BET : ARRAY OF HARDENING COEFFICIENTS FOR SAME TEMPERATURES WHERE
C           STRESS-STRAIN CURVES GIVEN
C
C   (IF LAWCRP.EQ.2.OR.LAWCRP.EQ.12)
C      (ENTER MTN OF THE FOLLOWING TWO LINE SETS)
C
C LINE  NCTP  TCUT  SNORM
C
C     NCTP  : NUMBER OF TEMPERATURES WHERE CREEP PROPERTIES SPECIFIED
C     TCUT  : CUTOFF TEMPERATURE BELOW WHICH NO CREEP OCCURS
C     SNORM : NORMALIZING STRESS (IN PSI) FOR THE FOLLOWING CREEP PROPERTIES
C
C     (ENTER NCTP OF THE FOLLOWING LINE)
C
C LINE  TEMP  Q  R  STRCUT
C
C     TEMP   : TEMPERATURE WHERE THESE CREEP PROPERTIES APPLY
C     Q      : CREEP PROPERTY
C     R      : CREEP PROPERTY  (CREEP STRAIN = Q * STRESS ** R)
C     STRCUT : CUTOFF STRESS (IN PSI) BELOW WHICH NO CREEP OCCURS
C
C-----------------------------------------------------------
C
C  VI TIME AND TIME INCREMENTING CONTROL INPUT
C
C LINE  N2M  TCRP  TINIT  ECMAX  SIGMAX  ERMAX  DELMIN  DELMUL
C
C   N2M    : NUMBER OF EQUAL TIME STEPS IN THIS LOAD CASE. IF N2M=0
C            DYNAMIC TIME INCREMENTING WILL BE USED
C   TCRP   : TOTAL TIME IN THIS LOAD CASE ( SECONDS ).
C   TINIT  : INITIAL TIME STEP. IF THIS IS NOT THE FIRST LOAD CASE AND
C     0.0 IS INPUT, .5 TIMES THE LAST CALCULATED TIME STEP OF THE
C     PREVIOUS LOAD CASE IS USED. IF THIS IS THE FIRST LOAD CASE OR
C     IF THIS IS THE FIRST LOAD CASE OF A RESTART AND 0.0 IS INPUT,
C     DELMIN IS USED.
C   ECMAX : MAXIMUM INELASTIC STRAIN INCREMENT DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS .000100.
C   SIGMAX : MAXIMUM CHANGE IN STRESS DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS 1000 PSI.
C   ERMAX : MAXIMUM ESTIMATED INTEGRATION ERROR ALLOWED IN ANY TIME STEP.
C     DEFAULT IS .01  ( 1% ).
C   DELMIN : MINIMUM ALLOWABLE TIME STEP. DEFAULT VALUE IS .001 TIMES
C     TCRP.
C   DELMUL : MAXIMUM MULTIPLIER ALLOWED ON CURRENT TIME STEP IN
C     CALCULATING THE NEXT TIME STEP. DEFAULT = 1.5
C
C
C-----------------------------------------------------------------------
C  VII CONVERGENCE CRITERIA
C
C LINE  IDM  ICON  DELSIG  DELEPS
C
C   IDM    : MATERIAL NUMBER
C   ICON   : CONVERGENCE CRITERIA CODE
C            = 1 CHANGE IN EFFECTIVE STRESS FOR SUBSEQUENT ITERATIONS
C                MUST BE LESS THAN DELSIG
C            = 2 CHANGE IN EFFECTIVE INELASTIC STRAIN INCREMENT FOR
C                 SUBSEQUENT ITERATIONS MUST BE LESS THAN DELEPS DELEPS
```

```
C             = 3 BOTH CONDITIONS 1 AND 2 MUST BE SATISFIED. DEFAULT = 3
C   DELSIG : CONVERGENCE TOLERANCE ON EFFECTIVE STRESS.
C            DEFAULT = .01*SIGMAX
C   DELEPS : CONVERGENCE TOLERANCE ON EFFECTIVE INELASTIC STRAIN INCREMENT.
C            DEFAULT = .01*ECMAX
C
C
C-----------------------------------------------------------------
C  VIII INITIAL CONSTRAINED DISPLACEMENTS
C
C LINE  NDB
C
C   NDB = NUMBER OF CONSTRAINED DISPLACEMENT SPECIFICATIONS
C
C ( ENTER NDB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C   N     : NODE NUMBER
C   IDIR  : DIRECTION CONSTRAINED
C         = 1 X CONSTRAINED    ( ANY COMBINATION OF CODES,
C         = 2 Y CONSTRAINED      i.e. 12, 13, OR 123
C         = 3 Z CONSTRAINED      MAY ALSO BE USED)
C   VALUE : NUMERICAL VALUE OF CONSTRAINED DISPLACEMENT. DEFAULT IS 0.0
C   NEND  : LAST NODE NUMBER HAVING THIS CONSTRAINT. IF OMITTED N IS ASSUMED
C   NINC  : INCREMENT TO BE USED FOR CONSTRAINT GENERATION FROM N TO NEND
C           DEFAULT IS 1. ( OPTIONAL )
C
C
C-----------------------------------------------------------------
C  IX INITIAL NODAL APPLIED FORCES
C
C LINE  NFB
C
C   NFB : NUMBER OF APPLIED NODAL FORCE SPECIFICATIONS
C
C ( ENTER NFB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C   N    : NODE NUMBER
C   IDIR : DIRECTION OF APPLIED FORCE
C        = 1 X DIRECTION
C         = 2 Y DIRECTION
C         = 3 Z DIRECTION
C   VALUE : NUMERICAL VALUE OF FORCE TO BE APPLIED ( LBS )
C   NEND  : LAST NODE HAVING THIS APPLIED FORCE. IF OMITTED N IS ASSUMED
C   NINC  : INCREMENT TO BE USED FOR FORCE GENERATION FROM N TO NEND.
C           DEFAULT IS 1. ( OPTIONAL )
C
C
C-----------------------------------------------------------------
C  X INITIAL NODAL TEMPERATURES
C
C LINE  NTEMPS
C
C   NTEMPS : NUMBER OF TEMPERATURE INPUT LINES
C
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C
C   N     : NODE NUMBER
C   TEMP  : TEMPERATURE ( DEGREES F. )
C   ITYPE : INDICATOR FOR TYPE OF TEMPERATURE REVISION
C         = 0 CHANGE NODAL TEMPERATURE TO TEMP
C         = 1 INCREMENT NODAL TEMPERATURE BY TEMP
```

```
5690C    NEND : LAST NODE HAVING THE GIVEN TEMPERATURE. IF OMITTED N IS ASSUMED
5700C    NINC : INCREMENT TO BE USED ON TEMPERATURE GENERATION FROM N TO NEND.
5710C           IF OMITTED 1 IS ASSUMED
5720C
5730C
5740C------------------------------------------------------------------------
5750C  XI INITIAL ELEMENT PRESSURE LOADS
5760C
5770C LINE  NPL
5780C
5790C   NPL : NUMBER OF PRESSURE INPUT LINES
5800C
5810C ( ENTER NPL OF THE FOLLOWING LINES )
5820C
5830C LINE  NEL  IFACE  P1  P2  P3  P4  P5  P6  P7  P8  NEND  NINC
5840C
5850C   NEL   : ELEMENT NUMBER
5860C   IFACE : FACE NUMBER ( IFACE.GE.1.AND.IFACE.LE.6 )
5870C   P1    : PRESSURE AT NODE 1 ( PSI )
5880C   P2, P3, P4, P5, P6, P7, P8 : PRESSURES AT NODES 2,....,8 ON THE FACE.
5890C                                IF OMITTED THEY ARE SET EQUAL TO P1.
5900            (POSITIVE PRESSURES INDUCE COMPRESSION IN THE ELEMENT )
5910C   NEND  : LAST ELEMENT HAVING THIS PRESSURE LOADING. IF OMITTED NEL IS
5920C           ASSUMED
5930C   NINC  : INCREMENT TO BE USED FOR ELEMENT PRESSURE GENERATION FROM
5940C           NEL TO NEND. IF OMITTED NEL IS ASSUMED
5950C
5960C
5970C------------------------------------------------------------------------
5980C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
5990C
6000C********************************************************************
6010C    XII.1 LOAD CASE CONTROL CARD
6020C
6030C LINE  RPM  IAXIS  IACC  NTI
6040C
6050C********************************************************************
6060C    XII.2 ACCELERATION SPECIFICATIONS FOR INERTIA OR GRAVITY LOADS
6070C
6080C LINE  ACCELX  ACCELY  ACCELZ
6090C
6100C
6110C------------------------------------------------------------------------
6120C  XIII NODAL CONSTRAINED DISPLACEMENTS
6130C
6140C LINE  NDB
6150C
6160C ( ENTER NDB OF THE FOLLOWING LINES )
6170C
6180C LINE  N  IDIR  VALUE  NEND  NINC
6190C
6200C
6210C------------------------------------------------------------------------
6220C XIV NODAL APPLIED FORCES
6230C
6240C LINE  NFB
6250C
6260C ( ENTER NFB OF THE FOLLOWING LINES )
6270C
6280C LINE  N  IDIR  VALUE  NEND  NINC
6290C
6300C
6310C------------------------------------------------------------------------
6320C XV NODAL TEMPERATURES
6330C
6340C LINE NTEMPS
6350C
```

```
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C
C
C-----------------------------------------------------------------
C XVI ELEMENT PRESSURE LOADS
C
C LINE NPL
C
C ( ENTER NPL OF THE FOLLOWING LINES )
C
C LINE  NEL  IFACE  P1  P2  P3  P4  P5  P6  P7  P8  NEND  NINC
```

```
C                ORGANIZATION OF INPUT
C         FOR T3CYAN WITH SIMPLE PLASTICITY AND SIMPLE CREEP
C
C  I HEADING AND CONTROL INFORMATION
C    I.1 TITLE CARD
C    I.2 PROBLEM SIZING
C    I.3 ANALYSIS AND RESTART OPTIONS
C    I.4 EQUATION NUMBERING AND BANDING OPTIONS
C
C  II NODE COORDINATES AND TRANSFORMATIONS
C    II.1 NODE COORDINATES
C    II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
C
C  III ELEMENT DEFINITION
C    III.1 HEADER LINE FOR ELEMENT
C    III.2 20 NODED SOLID DEFINITION
C
C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
C    IV.1 LOAD CASE CONTROL CARD
C    IV.2 ACCELERATION SPECIFICATIONS
C
C  V MATERIAL PHYSICAL PROPERTIES
C    V.1 MATERIAL PHYSICAL PROPERTIES
C      V.1.1 ISOTROPIC ELASTIC PROPERTIES
C      V.1.2 ORTHOTROPIC ELASTIC PROPERTIES
C    V.2 ORTHOTROPIC AXES ORIENTATION TABLE
C    V.3 INELASTIC MATERIAL CHARACTERIZATION
C
C  VI TIME AND TIME INCREMENTING CONTROL
C
C  VII CONVERGENCE CRITERIA
C
C  VIII INITIAL NODAL CONSTRAINED DISPLACEMENTS
C
C  IX INITIAL NODAL APPLIED FORCES
C
C  X INITIAL NODAL TEMPERATURES
C
C  XI INITIAL ELEMENT PRESSURE LOADS
C
C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
C    XII.1 LOAD CASE CONTROL CARD
C    XII.2 ACCELERATION SPECIFICATION
C
C  XIII FINAL NODAL CONSTRAINED DISPLACEMENTS
C
C  XIV FINAL NODAL APPLIED FORCES
C
C  XV FINAL NODAL TEMPERATURES
C
C  XVI FINAL ELEMENT PRESSURE LOADS
C
C
C-----------------------------------------------------------------
C  I HEADING AND CONTROL INFORMATION
C
C    I.1 TITLE CARD
C
C LINE  ITITLE
C
C   ITITLE = ANY 1 TO 72 CHARACTER TITLE FOR THE ANALYSIS
C
C*****************************************************************
C    I.2 PROBLEM SIZING
C
C LINE  NUMNP  NM  IT  RTEM  NLC
C
```

```
C    NUMNP  : NUMBER OF STRUCTURAL NODES ( ENTER AS A NEGATIVE NUMBER FOR
C             TIMING SUMMARY )
C    NM     : NUMBER OF DIFFERENT MATERIALS ( MAXIMUM=3 )
C    IT     : THERMAL STRESS OPTION
C            = 0  INCLUDE THERMAL LOADS
C            = 1  IGNORE THERMAL LOADS
C    RTEM   : REFERENCE TEMPERATURE ( DEGREES F )
C    NLC    : NUMBER OF LOAD CASES
C
C*****************************************************************
C     I.3 ANALYSIS AND RESTART OPTIONS
C
C LINE  LAWCRP  NOUT  NRESTA  INREST  MASSCD
C
C    LAWCRP : TYPE OF INELASTIC ANALYSIS
C             = 0  ELASTIC ANALYSIS
C             = 1  HAISLER-ALLEN PLASTICITY
C             = 2  SECONDARY CREEP MODEL
C             =12  PLASTICITY AND CREEP COMBINED
C    NOUT   : OUTPUT FILE CREATION OPTION
C             = 0  DO NOT CREATE OUTPUT FILE
C             = 1  CREATE OUTPUT FILE
C    NRESTA : RESTART OPTION
C             = 0  THIS IS NOT A RESTART RUN
C             > 0  INPUT THE LOAD CASE FROM WHICH THE RESTART IS TO PROCEED
C    NOTE:(OUTPUT FROM THIS CASE MUST HAVE BEEN PREVIOUSLY PUT ON AN OUTPUT
C      FILE.  THE FIRST NEW LOAD CASE WILL BE LABELED AS NRESTA + 1.)
C
C    INREST : LOAD CASE NUMBER IN THE CURRENT INPUT FILE WHICH BECOMES
C             THE FIRST NEW LOAD CASE TO BE SOLVED WHEN RESTARTING, WHERE
C             ( 1.LE.INREST.LE.NLC ). IF 0 IS INPUT INREST = 1 IS ASSUMED
C
C    MASSCD : MASS MATRIX FLAG
C             = 0  DO NOT CREATE MASS MATRIX
C             = 1  CREATE LUMPED MASS MATRIX
C             = 2  CREATE CONSISTENT MASS MATRIX
C
C*****************************************************************
C     I.4 EQUATION NUMBERING AND BANDING OPTIONS
C
C LINE  N  IBAND  IPBAND
C
C    N      : KEY CODE
C             = 0 NO NUMBERING OR BANDING
C             = -1 ACTIVATE NUMBERING AND BANDING OPTION
C    IBAND  : BANDING OPTION
C             = 0 USE DEFAULT OPTION
C            = 1 ASSUME NODE NUMBER IS THE SAME AS MATRIX POSITION
C            = 2 ASSUME INPUT NODE ORDER DEFINES MATRIX POSITION
C    IPBAND : PRINTOUT OPTION
C            = 0 NO PRINTOUT OF EQUATION NUMBERS
C            = 1 PRINT OUT THE EQUATION NUMBER FOR EACH DEGREE OF FREEDOM
C
C
C-----------------------------------------------------------------
C  II NODE COORDINATES AND TRANSFORMATIONS
C
C*****************************************************************
C     II.1 NODE COORDINATES
C
C     (ENTER THE FOLLOWING LINE FOR EACH NODE)
C
C LINE  N  X  Y  Z
C
C    N : NODE NUMBER
C    X : NODE X COORDINATE
C    Y : NODE Y COORDINATE
```

```
C   Z : NODE Z COORDINATE
C
C  (AFTER THE LAST NODE ENTER THE FOLLOWING LINE)
C
C LINE  O  NSKEW
C   NSKEW : INDICATOR FOR LOCAL NODE COORDINATE SYSTEM TRANSFORMATION
C           = O NONE WILL BE INPUT
C           = 1 ONE OR MORE WILL BE INPUT
C
C*****************************************************************************
C    II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
C
C  IF(NSKEW.GT.O)
C     ( ENTER ALL NODES WITH LOCAL COORDINATE SYSTEM TRANSFORMATIONS)
C
C LINE  N  NO  NI  NJ
C LINE O
C
C   N  : NODE NUMBER HAVING A ROTATED LOCAL COORDINATE SYSTEM
C   NO : NODE ON LOCAL X AXIS
C   NI : NODE ON LOCAL X AXIS IN THE +X DIRECTION FROM NO
C   NJ : NODE IN LOCAL XY PLANE SUCH THAT THE +Z AXIS IS IN THE
C        DIRECTION OF THE VECTOR P3 = (NO-NI) X (NO-NJ). THE +Y AXIS IS
C        IN THE DIRECTION OF THE VECTOR P2 = P3 X (NO-NI)
C
C
C
C-----------------------------------------------------------------------------
C  III ELEMENT DEFINITION
C
C*****************************************************************************
C    III.1 HEADER LINE FOR ELEMENT DATA
C
C LINE  IPRINT
C   IPRINT : ELEMENT DATA EXTENDED PRINTOUT OPTION
C            = O NONE
C            = 1 DUMP VOLUMES AND DISTORTION PARAMETERS
C            = 2 ALSO DUMP ELEMENT STIFFNESSES
C            = 3 ALSO DUMP ELEMENT EQUIVALENT NODAL FORCES AND MASSES
C
C*****************************************************************************
C    III.2 20 NODED SOLID DEFINITION
C
C     (ENTER THE FOLLOWING LINES FOR EACH ELEMENT)
C
C LINE  NEL  N1  N2  N3  N4  N5  N6  N7  N8  N9  N10  N11  N12
C LINE  N13  N14  N15  N16  N17  N18  N19  N20  IMAT  IOR
C LINE NO  NP  NQ    ( OPTIONAL LINE, ENTER ONLY IF IOR = 100)
C
C LINE  O
C
C   NEL  : ELEMENT NUMBER
C   N1 ... N20 : NODES DEFINING THE ELEMENT  ( SEE FIG III.2 )
C   IMAT : MATERIAL NUMBER
C   IOR  : ORTHOTROPIC MATERIAL AXIS SYSTEM INDICATOR
C          = O  ISOTROPIC MATERIAL
C          (1.LE.IOR.LE.99) IOR IS ORIENTATION IDENTIFIER OF SYSTEM
C                           GIVEN UNDER MATERIAL DATA SECTION V.2
C          = 100 ORTHOTROPIC AXES DEFINED BY OPTIONAL LINE OF INPUT
C
C
C
C-----------------------------------------------------------------------------
C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
C
C*****************************************************************************
C    IV.1 LOAD CASE CONTROL CARD
```

```
C
C LINE  RPM  IAXIS  IACC  NPCI  NTI  ITHST
C
C    RPM    : ROTATIONAL SPEED IN RPM
C    IAXIS  : GLOBAL AXIS ABOUT WHICH STRUCTURE IS ROTATING
C             = 1 X AXIS
C             = 2 Y AXIS
C             = 3 Z AXIS
C    IACC   : INDICATOR FOR INPUT OF ACCELERATON LOADS IN GROUP IV.2
C             = 0 NO INPUT
C             = 1 ACCELERATION LOADS ARE INPUT
C    NPCI  : INDICATOR FOR A CHANGE IN MATERIAL PROPERTIES
C             = 0 NO CHANGE TO MATERIAL PROPERTIES
C             = 1 CHANGE ELASTIC PROPERTIES ( GROUPS V.1 AND V.2)
C             = 2 CHANGE INELASTIC PROPERTIES ( GROUP V.3 )
C             = 12 CHANGE ELASTIC AND INELASTIC PROPERTIES
C    NOTE: IF THIS IS FIRST LOAD CASE, PROGRAM SETS NPCI=12
C
C    NTI    : NUMBER OF NODAL TEMPERATURES WHICH ARE RESPECIFIED,
C             = 0 NO RESPECIFIED TEMPERATURES, STIFFNESS IS RECOMPUTED
C             = -1 NO RESPECIFIED TEMPERATURES, STIFFNESS IS NOT RECOMPUTED
C             > 0 NTI RESPECIFIED TEMPERATURES, STIFFNESS IS RECOMPUTED
C    ITHST  : TOTAL STRAIN PRINTOUT OPTION
C             = 0 INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C             = 1 DO NOT INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C
C***************************************************************
C     IV.2 ACCELERATION SPECIFICATION FOR INERTIAL OR GRAVITY LOADS
C
C     IF(IACC.GT.0)
C LINE  ACCELX  ACCELY  ACCELZ
C
C    ACCELX : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL X
C    ACCELY : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Y
C    ACCELZ : ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Z
C
C
C---------------------------------------------------------------
C  V MATERIAL PHYSICAL PROPERTIES
C
C***************************************************************
C     V.1 ELASTIC CONSTANTS
C
C    IF(NPCI.EQ.1.OR.NPCI.EQ.12)
C      (ENTER NM OF THE FOLLOWING LINES)
C
C LINE  MTN  NMT  DEN
C
C    MTN : MATERIAL NUMBER ( MTN.GE.1.AND.MTN.LE.NM )
C         (IF MATERIAL IS ISOTROPIC, INPUT MTN AS A NEGATIVE NUMBER
C          TO SIMPLIFY INPUT)
C    NMT : NUMBER OF TEMPERATURES AT WHICH ELASTIC PROPERTIES WILL BE
C          GIVEN FOR THIS MATERIAL
C    DEN : WEIGHT DENSITY OF THE MATERIAL ( POUNDS/IN**3 )
C
C###############################################################
C     V.1.1 ISOTROPIC MATERIAL
C
C    IF(MTN.LT.0)
C    ( ENTER NMT OF THESE LINES )
C
C LINE  TEMP  E  PR  AL
C
C    TEMP  : TEMPERATURE ( DEGREES F. )
C    E     : ELASTIC MODULUS ( 10**6 P.S.I. )
C    PR    : POISSON'S RATIO
C    AL    : MEAN COEFFICIENT OF THERMAL EXPANSION ( 10**-6 IN/IN-DEG. F. )
```

```
C
C##################################################################
C     V.1.2 ORTHOTROPIC MATERIAL
C
C    IF(MTN.GT.O)
C     (ENTER NMT OF THE FOLLOWING LINES)
C     (DIRECTIONS 1,2,3 CORRESPOND TO THE MATERIAL ORTHOTROPIC AXES X',Y',Z')
C
C LINE TEMP E11 E22 E33 NU12 NU13 NU23 G12 G23 G31 AL1 AL2 AL3
C
C    TEMP : TEMPERATURE AT WHICH PROPERTIES ARE GIVEN ( DEG. F. )
C    E11  : ELASTIC MODULUS IN THE 1 DIRECTION
C    E22  : ELASTIC MODULUS IN THE 2 DIRECTION
C    E33  : ELASTIC MODULUS IN THE 3 DIRECTION
C    NU12 : POISSON'S RATIO RELATING DIRECTIONS 1 AND 2
C    NU13 : POISSON'S RATIO RELATING DIRECTIONS 1 AND 3
C    NU23 : POISSON'S RATIO RELATING DIRECTIONS 2 AND 3
C    G12  : SHEAR MODULUS IN THE 1-2 PLANE
C    G23  : SHEAR MODULUS IN THE 2-3 PLANE
C    G31  : SHEAR MODULUS IN THE 3-1 PLANE
C    AL1  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 1 DIRECTION
C    AL2  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 2 DIRECTION
C    AL3  : MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 3 DIRECTION
C
C*****************************************************************
C     V.2 ORTHOTROPIC AXIS ORIENTATION TABLE
C
C LINE  NOR
C
C   NOR : NUMBER OF ORIENTATION SPECIFICATIONS
C         (O.LE.NOR.LE.10) i.e. MAX OF 10 ORIENTATION SYSTEMS ALLOWED
C      (NOTE: IF THE MATERIALS ARE ISOTROPIC OR THE ORTHOTROPIC AXES
C             COINCIDE WITH THE GLOBAL AXES, ENTER O)
C      ENTER O. ( MAXIMUM OF 10 SPECIFICATIONS ALLOWED )
C
C   IF(NOR.GT.O)
C  ( ENTER NOR LINES OF THE FOLLOWING )
C
C LINE  I  NO  NP  NQ
C
C   I             : ORIENTATION IDENTIFIER ( IOR IN ELEMENT INPUT )
C   NO, NP, NQ    : NODE NUMBERS IDENTIFYING ORTHOTROPIC AXES ( SEC. II.2 )
C
C*****************************************************************
C  V.3 INELASTIC MATERIAL PROPERTIES
C
C      (IF NPCI.EQ.2.OR.NPCI.EQ.12 )
C      (IF LAWCRP.EQ.1.OR.LAWCRP.EQ.12)
C      (ENTER MTN OF THE NEXT FOUR LINE SETS - MTN,SSTEMP,PPV)
C
C LINE  MTN  NPTS  NTM
C
C      MTN  : MATERIAL NUMBER
C      NPTS : NUMBER OF STRESS-STRAIN POINTS PER CURVE
C      NTM  : NUMBER OF TEMPERATURES FOR WHICH STRESS-STRAIN CURVES ARE GIVEN
C
C
C LINE  SSTEMP(1)  SSTEMP(2) ..... SSTEMP(NTM)
C
C      SSTEMP : ARRAY OF TEMPERATURES WHERE STRESS-STRAIN CURVES ARE GIVEN
C               (IN INCREASING ORDER)
C
C LINE  PPV(MTN,I,1,1)  PPV(MTN,I,1,2)
C LINE  PPV(MTN,I,2,1)  PPV(MTN,I,2,2)
C             .               .
C             .               .
C             .               .
```

```
C LINE  PPV(MTN,I,NPTS,1)  PPV(MTN,I,NPTS,2)
C
C     PPV(MTN,I,J,1) : STRESS VALUES IN INCREASING ORDER FROM J=1 TO NPTS
C                      FOR TEMPERATURE I AND MATERIAL MTN
C     PPV(MTN,I,J,2) : STRAIN VALUES IN INCREASING ORDER FROM J=1 TO NPTS
C                      FOR TEMPERATURE I AND MATERIAL MTN
C
C   (IF LAWCRP.EQ.2.OR.LAWCRP.EQ.12)
C      (ENTER MTN OF THE FOLLOWING TWO LINE SETS)
C
C LINE  NCTP  TCUT  SNORM
C
C     NCTP  : NUMBER OF TEMPERATURES WHERE CREEP PROPERTIES SPECIFIED
C     TCUT  : CUTOFF TEMPERATURE BELOW WHICH NO CREEP OCCURS
C     SNORM : NORMALIZING STRESS (IN PSI) FOR THE FOLLOWING CREEP PROPERTIES
C
C     (ENTER NCTP OF THE FOLLOWING LINE)
C
C LINE  TEMP  Q  R  STRCUT
C
C     TEMP   : TEMPERATURE WHERE THESE CREEP PROPERTIES APPLY
C     Q      : CREEP PROPERTY
C     R      : CREEP PROPERTY  (CREEP STRAIN = Q * STRESS ** R)
C     STRCUT : CUTOFF STRESS (IN PSI) BELOW WHICH NO CREEP OCCURS
C
C-----------------------------------------------------------
C
C VI TIME AND TIME INCREMENTING CONTROL INPUT
C
C LINE  N2M  TCRP  TINIT  ECMAX  SIGMAX  ERMAX  DELMIN  DELMUL
C
C   N2M    : NUMBER OF EQUAL TIME STEPS IN THIS LOAD CASE, IF N2M=0
C              DYNAMIC TIME INCREMENTING WILL BE USED
C   TCRP   : TOTAL TIME IN THIS LOAD CASE ( SECONDS ).
C   TINIT  : INITIAL TIME STEP. IF THIS IS NOT THE FIRST LOAD CASE AND
C     0.0 IS INPUT, .5 TIMES THE LAST CALCULATED TIME STEP OF THE
C     PREVIOUS LOAD CASE IS USED. IF THIS IS THE FIRST LOAD CASE OR
C     IF THIS IS THE FIRST LOAD CASE OF A RESTART AND 0.0 IS INPUT,
C     DELMIN IS USED.
C   ECMAX : MAXIMUM INELASTIC STRAIN INCREMENT DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS .000100.
C   SIGMAX : MAXIMUM CHANGE IN STRESS DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS 1000 PSI.
C   ERMAX : MAXIMUM ESTIMATED INTEGRATION ERROR ALLOWED IN ANY TIME STEP.
C     DEFAULT IS .01  ( 1% ).
C   DELMIN : MINIMUM ALLOWABLE TIME STEP. DEFAULT VALUE IS .001 TIMES
C     TCRP.
C   DELMUL : MAXIMUM MULTIPLIER ALLOWED ON CURRENT TIME STEP IN
C     CALCULATING THE NEXT TIME STEP. DEFAULT = 1.5
C
C
C-----------------------------------------------------------------------
C VII CONVERGENCE CRITERIA
C
C LINE  IDM  ICON  DELSIG  DELEPS
C
C   IDM    : MATERIAL NUMBER
C   ICON   : CONVERGENCE CRITERIA CODE
C            = 1 CHANGE IN EFFECTIVE STRESS FOR SUBSEQUENT ITERATIONS
C                MUST BE LESS THAN DELSIG
C            = 2 CHANGE IN EFFECTIVE INELASTIC STRAIN INCREMENT FOR
C                  SUBSEQUENT ITERATIONS MUST BE LESS THAN DELEPS DELEPS
C            = 3 BOTH CONDITIONS 1 AND 2 MUST BE SATISFIED, DEFAULT = 3
C   DELSIG : CONVERGENCE TOLERANCE ON EFFECTIVE STRESS.
C            DEFAULT = .01*SIGMAX
C   DELEPS : CONVERGENCE TOLERANCE ON EFFECTIVE INELASTIC STRAIN INCREMENT.
C            DEFAULT = .01*ECMAX
```

```
C
C
C-----------------------------------------------------------------
C  VIII INITIAL CONSTRAINED DISPLACEMENTS
C
C LINE  NDB
C
C   NDB = NUMBER OF CONSTRAINED DISPLACEMENT SPECIFICATIONS
C
C ( ENTER NDB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C   N     : NODE NUMBER
C   IDIR  : DIRECTION CONSTRAINED
C         = 1 X CONSTRAINED   ( ANY COMBINATION OF CODES,
C         = 2 Y CONSTRAINED     i.e. 12, 13, OR 123
C         = 3 Z CONSTRAINED     MAY ALSO BE USED)
C   VALUE : NUMERICAL VALUE OF CONSTRAINED DISPLACEMENT. DEFAULT IS 0.0
C   NEND  : LAST NODE NUMBER HAVING THIS CONSTRAINT. IF OMITTED N IS ASSUMED
C   NINC  : INCREMENT TO BE USED FOR CONSTRAINT GENERATION FROM N TO NEND
C           DEFAULT IS 1. ( OPTIONAL )
C
C
C-----------------------------------------------------------------
C  IX INITIAL NODAL APPLIED FORCES
C
C LINE  NFB
C
C   NFB : NUMBER OF APPLIED NODAL FORCE SPECIFICATIONS
C
C ( ENTER NFB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C   N    : NODE NUMBER
C   IDIR : DIRECTION OF APPLIED FORCE
C        = 1 X DIRECTION
C        = 2 Y DIRECTION
C        = 3 Z DIRECTION
C   VALUE : NUMERICAL VALUE OF FORCE TO BE APPLIED ( LBS )
C   NEND  : LAST NODE HAVING THIS APPLIED FORCE. IF OMITTED N IS ASSUMED
C   NINC  : INCREMENT TO BE USED FOR FORCE GENERATION FROM N TO NEND.
C           DEFAULT IS 1. ( OPTIONAL )
C
C
C-----------------------------------------------------------------
C  X INITIAL NODAL TEMPERATURES
C
C LINE  NTEMPS
C
C   NTEMPS : NUMBER OF TEMPERATURE INPUT LINES
C
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C
C   N     : NODE NUMBER
C   TEMP  : TEMPERATURE ( DEGREES F. )
C   ITYPE : INDICATOR FOR TYPE OF TEMPERATURE REVISION
C         = 0 CHANGE NODAL TEMPERATURE TO TEMP
C         = 1 INCREMENT NODAL TEMPERATURE BY TEMP
C   NEND : LAST NODE HAVING THE GIVEN TEMPERATURE. IF OMITTED N IS ASSUMED
C   NINC : INCREMENT TO BE USED ON TEMPERATURE GENERATION FROM N TO NEND.
C          IF OMITTED 1 IS ASSUMED
C
C
```

```
C-----------------------------------------------------------------
C  XI INITIAL ELEMENT PRESSURE LOADS
C
C LINE  NPL
C
C   NPL  : NUMBER OF PRESSURE INPUT LINES
C
C ( ENTER NPL OF THE FOLLOWING LINES )
C
C LINE  NEL  IFACE  P1  P2  P3  P4  P5  P6  P7  P8  NEND  NINC
C
C   NEL    : ELEMENT NUMBER
C   IFACE  : FACE NUMBER ( IFACE.GE.1.AND.IFACE.LE.6 )
C   P1     : PRESSURE AT NODE 1 ( PSI )
C   P2, P3, P4, P5, P6, P7, P8 : PRESSURES AT NODES 2,....8 ON THE FACE.
C                                IF OMITTED THEY ARE SET EQUAL TO P1.
            (POSITIVE PRESSURES INDUCE COMPRESSION IN THE ELEMENT )
C   NEND   : LAST ELEMENT HAVING THIS PRESSURE LOADING. IF OMITTED NEL IS
C            ASSUMED
C   NINC   : INCREMENT TO BE USED FOR ELEMENT PRESSURE GENERATION FROM
C            NEL TO NEND. IF OMITTED NEL IS ASSUMED
C
C
C-----------------------------------------------------------------
C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
C
C*****************************************************************
C     XII.1 LOAD CASE CONTROL CARD
C
C LINE  RPM  IAXIS  IACC  NTI
C
C*****************************************************************
C     XII.2 ACCELERATION SPECIFICATIONS FOR INERTIA OR GRAVITY LOADS
C
C LINE  ACCELX  ACCELY  ACCELZ
C
C
C-----------------------------------------------------------------
C  XIII NODAL CONSTRAINED DISPLACEMENTS
C
C LINE  NDB
C
C ( ENTER NDB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C
C-----------------------------------------------------------------
C XIV NODAL APPLIED FORCES
C
C LINE  NFB
C
C ( ENTER NFB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C
C-----------------------------------------------------------------
C XV NODAL TEMPERATURES
C
C LINE NTEMPS
C
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C
C
```

```
C------------------------------------------------------------------
C XVI ELEMENT PRESSURE LOADS
C
C LINE NPL
C
C ( ENTER NPL OF THE FOLLOWING LINES )
C
C LINE  NEL  IFACE  P1  P2  P3  P4  P5  P6  P7  P8  NEND  NINC
```

```
C                ORGANIZATION OF INPUT
C
C  I HEADING AND CONTROL INFORMATION
C    I.1 TITLE CARD
C    I.2 PROBLEM SIZING
C    I.3 ANALYSIS AND RESTART OPTIONS
C    I.4 EQUATION NUMBERING AND BANDING OPTIONS
C
C  II NODE COORDINATES AND TRANSFORMATIONS
C    II.1 NODE COORDINATES
C    II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
C
C  III ELEMENT DEFINITION
C    III.1 HEADER LINE FOR ELEMENT
C    III.2 20 NODED SOLID DEFINITION
C
C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
C    IV.1 LOAD CASE CONTROL CARD
C    IV.2 ACCELERATION SPECIFICATIONS
C
C  V MATERIAL PHYSICAL PROPERTIES
C    V.1 MATERIAL PHYSICAL PROPERTIES
C      V.1.1 ISOTROPIC ELASTIC PROPERTIES
C      V.1.2 ORTHOTROPIC ELASTIC PROPERTIES
C    V.2 ORTHOTROPIC AXES ORIENTATION TABLE
C    V.3 INELASTIC MATERIAL CHARACTERIZATION
C
C  VI TIME AND TIME INCREMENTING CONTROL
C
C  VII CONVERGENCE CRITERIA
C
C  VIII INITIAL NODAL CONSTRAINED DISPLACEMENTS
C
C  IX INITIAL NODAL APPLIED FORCES
C
C  X INITIAL NODAL TEMPERATURES
C
C  XI INITIAL ELEMENT PRESSURE LOADS
C
C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
C    XII.1 LOAD CASE CONTROL CARD
C    XII.2 ACCELERATION SPECIFICATION
C
C  XIII FINAL NODAL CONSTRAINED DISPLACEMENTS
C
C  XIV FINAL NODAL APPLIED FORCES
C
C  XV FINAL NODAL TEMPERATURES
C
C  XVI FINAL ELEMENT PRESSURE LOADS
C
C
        END
```

```
C  I HEADING AND CONTROL INFORMATION
C
C     I.1 TITLE CARD
C
C LINE  ITITLE
C    ITITLE = ANY 1 TO 72 CHARACTER TITLE FOR THE ANALYSIS
C
C     I.2 PROBLEM SIZING
C
C LINE  NUMNP  NM  IT  RTEM  NLC
C    NUMNP = NUMBER OF STRUCTURAL NODES ( ENTER AS A NEGATIVE NUMBER FOR
C      TIMING SUMMARY )
C    NM = NUMBER OF DIFFERENT MATERIALS ( MAXIMUM=3 )
C    IT = THERMAL STRESS OPTION
C      = 0  INCLUDE THERMAL LOADS
C      = 1  IGNORE THERMAL LOADS
C    RTEM = REFERENCE TEMPERATURE ( DEGREES F )
C
C     I.3 ANALYSIS AND RESTART OPTIONS
C
C LINE  LAWCRP  NOUT  NRESTA  INREST
C    LAWCRP = TYPE OF INELASTIC ANALYSIS
C      = 0  ELASTIC ANALYSIS
C      = 1  ISOTHERMAL BODNER MODEL
C    NOUT = OUTPUT FILE CREATION OPTION
C      = 0  DO NOT CREATE OUTPUT FILE
C      = 1  CREATE OUTPUT FILE
C    NRESTA = RESTART OPTION
C      = 0  THIS IS NOT A RESTART RUN
C      > 0  INPUT THE LOAD CASE FROM WHICH THE RESTART IS TO PROCEED
C      ( OUTPUT FROM THIS CASE MUST HAVE BEEN PREVIOUSLY PUT ON AN OUTPUT
C      FILE ). THE FIRST NEW LOAD CASE WILL BE LABELED AS NRESTA + 1.
C    INREST = LOAD CASE NUMBER IN THE CURRENT INPUT FILE WHICH BECOMES
C      THE FIRST NEW LOAD CASE TO BE SOLVED WHEN RESTARTING, WHERE
C      ( 1.LE.INREST.LE.NLC ). IF 0 IS INPUT INREST = 1 IS ASSUMED
C
C     I.4 EQUATION NUMBERING AND BANDING OPTIONS
C
C LINE  N  IBAND  IPBAND
C    N = KEY CODE, ENTER -1 TO ACTIVATE THIS OPTION
C    IBAND = BANDING OPTION
C      0 = USE DEFAULT OPTION
C      1 = ASSUME NODE NUMBER IS THE SAME AS MATRIX POSITION
C      2 = ASSUME INPUT NODE ORDER DEFINES MATRIX POSITION
C    IPBAND = PRINTOUT OPTION
C      0 = NO PRINTOUT OF EQUATION NUMBERS
C      1 = PRINT OUT THE EQUATION NUMBER FOR EACH DEGREE OF FREEDOM
C
C
          END
```

```
C  II NODE COORDINATES AND TRANSFORMATIONS
C
C     II.1 NODE COORDINATES
C
C LINE   N   X   Y   Z
C   N = NODE NUMBER
C   X = NODE X COORDINATE
C   Y = NODE Y COORDINATE
C   Z = NODE Z COORDINATE
C
C  AFTER THE LAST NODE ENTER THE FOLLOWING LINE
C
C LINE   0   NSKEW
C   NSKEW = INDICATOR FOR LOCAL NODE COORDINATE SYSTEM TRANSFORMATION
C     0 = NONE WILL BE INPUT
C     1 = ONE OR MORE WILL BE INPUT
C
C     II.2 LOCAL NODE COORDINATE SYSTEM TRANSFORMATIONS
C
C     ( IF NSKEW > 0, ENTER ALL NODES WITH LOCAL COORDINATE SYSTEM
C     TRANSFORMATIONS )
C
C LINE   N   NO   NI   NJ
C   N = NODE NUMBER HAVING A ROTATED LOCAL COORDINATE SYSTEM
C   NO = NODE ON LOCAL X AXIS
C   NI = NODE ON LOCAL X AXIS IN THE +X DIRECTION FROM NO
C   NJ = NODE IN LOCAL XY PLANE SUCH THAT THE +Z AXIS IS IN THE
C     DIRECTION OF THE VECTOR P3 = (NO-NI) X (NO-NJ). THE +Y AXIS IS
C     IN THE DIRECTION OF THE VECTOR P2 = P3 X (NO-NI)
C
C  ( AFTER THE LAST NODE HAVING A LOCAL COORDINATE SYSTEM ENTER A LINE
C     WITH N = 0 ) I.E.
C
C LINE 0
C
C
       END
```

```
C  III ELEMENT DEFINITION
C
C    III.1 HEADER LINE FOR ELEMENT DATA
C
C LINE  IPRINT(2)
C   IPRINT(2) = ELEMENT DATA EXTENDED PRINTOUT OPTION
C     0 = NONE
C     1 = DUMP VOLUMES AND DISTORTION PARAMETERS
C     2 = ALSO DUMP ELEMENT STIFFNESSES
C     3 = ALSO DUMP ELEMENT EQUIVALENT NODAL FORCES AND MASSES
C
C    III.2 20 NODED SOLID DEFINITION
C
C LINE  NEL  N1  N2  N3  N4  N5  N6  N7  N8  N9  N10  N11  N12
C LINE  N13  N14  N15  N16  N17  N18  N19  N20  IMAT  IOR
C LINE NO  NP  NQ    ( OPTIONAL LINE, ENTER ONLY IF IOR = 100)
C   NEL = ELEMENT NUMBER
C   N1 ... N20 = NODES DEFINING THE ELEMENT  ( SEE FIG III.2 )
C   IMAT = MATERIAL NUMBER
C   IOR = ORTHOTROPIC MATERIAL AXIS SYSTEM INDICATOR
C     IOR = 0  ISOTROPIC MATERIAL
C     IOR.GE.1.AND.IOR.LE.99 ORTHOTROPIC AXES SPECIFIED BY THE TABLE
C     GIVEN UNDER MATERIAL DATA SECTION V.2
C     IOR=100 ORTHOTROPIC AXES DEFINED BY OPTIONAL LINE OF INPUT
C
C  ( AFTER THE LAST ELEMENT, ENTER THE FOLLOWING LINE )
C
C LINE  0
C
C
      END
```

```
C  IV LOAD CASE INFORMATION, INITIAL CONDITIONS
C
C     IV.1 LOAD CASE CONTROL CARD
C
C LINE  RPM  IAXIS  IACC  NPCI  NTI  ITHST
C    RPM = ROTATIONAL SPEED IN RPM
C    IAXIS = GLOBAL AXIS ABOUT WHICH STRUCTURE IS ROTATING
C      1 = X AXIS
C      2 = Y AXIS
C      3 = Z AXIS
C    IACC = INDICATOR FOR INPUT OF ACCELERATON LOADS IN GROUP IV.2
C      0 = NO INPUT
C      1 = INPUT
C    NPCI = INDICATOR FOR A CHANGE IN MATERIAL PROPERTIES
C      0 = NO CHANGE TO MATERIAL PROPERTIES
C      1 = CHNAGE ELASTIC PROPERTIES ( GROUPS V.1 AND V.2)
C      2 = CHANGE INELASTIC PROPERTIES ( GROUP V.3 )
C      12 = CHANGE ELASTIC AND INELASTIC PROPERTIES
C    NTI = NUMBER OF NODAL TEMPERATURES WHICH ARE RESPECIFIED.
C      0 = NO RESPECIFIED TEMPERATURES BUT STIFFNESS IS RECOMPUTED
C      -1 = NO RESPECIFIED TEMPERATURES AND STIFFNESS IS NOT RECOMPUTED
C    ITHST = TOTAL STRAIN PRINTOUT OPTION
C      0 = INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C      1 = DO NOT INCLUDE THERMAL STRAINS IN TOTAL STRAIN PRINTOUT
C
C     IV.2 ACCELERATION SPECIFICATION FOR INERTIAL OR GRAVITY LOADS
C
C      ( ENTER ONLY IF IACC > 0 )
C
C LINE  ACCELX  ACCELY  ACCELZ
C    ACCELX = ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL X
C    ACCELY = ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Y
C    ACCELZ = ACCELERATION OF STRUCTURE ( IN/SEC**2 ) IN GLOBAL Z
C
C
       END
```

```
C  V MATERIAL PHYSICAL PROPERTIES
C
C     V.1 ELASTIC CONSTANTS ( ENTER NM GROUPS )
C
C LINE  MTN  NMT  DEN
C   MTN = MATERIAL NUMBER ( MTN.GE.1.AND.MTN.LE.NM )
C     (IF MATERIAL IS ISOTROPIC, INPUT MTN AS A NEGATIVE NUMBER TO
C     SIMPLIFY INPUT)
C   NMT = NUMBER OF TEMPERATURES AT WHICH ELASTIC PROPERTIES WILL BE
C     GIVEN FOR THIS MATERIAL
C   DEN = WEIGHT DENSITY OF THE MATERIAL ( POUNDS/IN**3 )
C
C     V.1.1 ISOTROPIC MATERIAL
C
C  ( IF MTN < 0, ENTER NMT OF THESE LINES )
C
C LINE  TEMP  E  PR  AL
C   TEMP = TEMPERATURE ( DEGREES F. )
C   E = ELASTIC MODULUS ( 10**6 P.S.I. )
C   PR = POISSON'S RATIO
C   AL = MEAN COEFFICIENT OF THERMAL EXPANSION ( 10**-6 IN/IN-DEG. F. )
C
C     V.1.2 ORTHOTROPIC MATERIAL
C
C  ( IF MTN > 0, ENTER NMT OF THESE LINES. THE DIRECTIONS 1,2,3
C  CORRESPOND TO THE MATERIAL ORTHOTROPIC AXES X',Y',Z'. )
C
C LINE TEMP E11 E22 E33 NU12 NU13 NU23 G12 G23 G31 AL1 AL2 AL3
C   TEMP = TEMPERATURE AT WHICH PROPERTIES ARE GIVEN ( DEG. F. )
C   E11 = ELASTIC MODULUS IN THE 1 DIRECTION
C   E22 = ELASTIC MODULUS IN THE 2 DIRECTION
C   E33 = ELASTIC MODULUS IN THE 3 DIRECTION
C   NU12 = POISSON'S RATIO RELATING DIRECTIONS 1 AND 2
C   NU13 = POISSON'S RATIO RELATING DIRECTIONS 1 AND 3
C   NU23 = POISSON'S RATIO RELATING DIRECTIONS 2 AND 3
C   G12 = SHEAR MODULUS IN THE 1-2 PLANE
C   G23 = SHEAR MODULUS IN THE 2-3 PLANE
C   G31 = SHEAR MODULUS IN THE 3-1 PLANE
C   AL1 = MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 1 DIRECTION
C   AL2 = MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 2 DIRECTION
C   AL3 = MEAN COEFFICIENT OF THERMAL EXPANSION IN THE 3 DIRECTION
C
C     V.2 ORTHOTROPIC AXIS ORIENTATION TABLE
C
C LINE  NOR
C   NOR = NUMBER OF ORIENTATION SPECIFICATIONS. IF THE MATERIALS ARE
C     ISOTROPIC OR THE ORTHOTROPIC AXES COINCIDE WITH THE GLOBAL AXES
C     ENTER 0. ( MAXIMUM OF 10 SPECIFICATIONS ALLOWED )
C
C  ( ENTER NOR LINES OF THE FOLLOWING )
C
C LINE  I  NO  NP  NQ
C   I = ORIENTATION IDENTIFIER ( IOR IN ELEMENT INPUT )
C   NO, NP, NQ = NODE NUMBERS IDENTIFYING ORTHOTROPIC AXES ( SEC. II.2 )
C
C  V.3 INELASTIC MATERIAL PROPERTIES
C
C LINE  MTN  NCTEM
C   MTN = MATERIAL NUMBER
C   NCTEM = NUMBER OF TEMPERATURES AT WHICH INELASTIC MATERIAL PROPERTIES
C     WILL BE GIVEN FOR THIS MATERIAL
C
C ( ENTER NCTEM OF THE FOLLOWING LINES )
C
C LINE  TEMP  D  AN  ZO  Z1  Z2  AM  A  R
C  TEMP = TEMPERATURE IN DEGREES F.
C  D, AN, ZO, Z1, Z2, AM, A, R = MATERIAL PARAMETERS REQUIRED FOR BODNER'S
```

```
C     INELASTIC CONSTITUTIVE MODEL AT THIS TEMPERATURE
C
C
      END
```

```
C  VI TIME AND TIME INCREMENTING CONTROL INPUT
C
C LINE  N2M  TCRP  TINIT  ECMAX  SIGMAX  ERMAX  DELMIN  DELMUL
C   N2M = NUMBER OF EQUAL TIME STEPS IN THIS LOAD CASE. IF N2M=0
C     DYNAMIC TIME INCREMENTING WILL BE USED
C   TCRP = TOTAL TIME IN THIS LOAD CASE ( SECONDS ).
C   TINIT = INITIAL TIME STEP. IF THIS IS NOT THE FIRST LOAD CASE AND
C     0.0 IS INPUT, .5 TIMES THE LAST CALCULATED TIME STEP OF THE
C     PREVIOUS LOAD CASE IS USED. IF THIS IS THE FIRST LOAD CASE OR
C     IF THIS IS THE FIRST LOAD CASE OF A RESTART AND 0.0 IS INPUT,
C     DELMIN IS USED.
C   ECMAX = MAXIMUM INELASTIC STRAIN INCREMENT DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS .000100.
C   SIGMAX = MAXIMUM CHANGE IN STRESS DESIRED IN ANY TIME STEP.
C     DEFAULT VALUE IS 1000 PSI.
C   ERMAX = MAXIMUM ESTIMATED INTEGRATION ERROR ALLOWED IN ANY TIME STEP.
C     DEFAULT IS .01  ( 1% ).
C   DELMIN = MINIMUM ALLOWABLE TIME STEP. DEFAULT VALUE IS .001 TIMES
C     TCRP.
C   DELMUL = MAXIMUM MULTIPLIER ALLOWED ON CURRENT TIME STEP IN
C     CALCULATING THE NEXT TIME STEP. DEFAULT = 1.5
C
C
      END
```

```
C  VII CONVERGENCE CRITERIA
C
C LINE  IDM  ICON  DELSIG  DELEPS
C   IDM = MATERIAL NUMBER
C   ICON = CONVERGENCE CRITERIA CODE
C     = 1 CHANGE IN EFFECTIVE SRESS FOR SUBSEQUENT ITERATIONS < DELSIG
C     = 2 CHANGE IN EFFECTIVE INELASTIC STRAIN INCREMENT FOR SUBSEQUENT
C     ITERATIONS < DELEPS
C     = 3 BOTH CONDITIONS 1 AND 2 MUST BE SATISFIED. DEFAULT = 3
C   DELSIG = CONVERGENCE TOLERANCE ON EFFECTIVE STRESS.
C     DEFAULT = .01*SIGMAX
C   DELEPS = CONVERGENCE TOLERANCE ON EFFECTIVE INELASTIC STRAIN INCREMENT.
C     DEFAULT = .01*ECMAX
C
C
      END
```

```
C  VIII INITIAL CONSTRAINED DISPLACEMENTS
C
C LINE  NDB
C   NDB = NUMBER OF CONSTRAINED DISPLACEMENT SPECIFICATIONS
C
C ( ENTER NDB OF TH FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C   N = NODE NUMBER
C   IDIR = DIRECTION CONSTRAINED
C     1 = X        ( ANY COMBINATION OF CODES, i.e. 12, 13 OR 123 MAY
C     2 = Y        ALSO BE USED)
C     3 = Z
C   VALUE = NUMERICAL VALUE OF CONSTRAINED DISPLACEMENT. DEFAULT IS 0.0
C   NEND = LAST NODE NUMBER HAVING THIS CONSTRAINT. IF OMITTED N IS ASSUMED
C   NINC = INCREMENT TO BE USED FOR CONSTRAINT GENERATION FROM N TO NEND
C     DEFAULT IS 1. ( OPTIONAL )
C
C
      END
```

```
C  IX INITIAL NODAL APPLIED FORCES
C
C LINE  NFB
C   NFB = NUMBER OF APPLIED NODAL FORCE SPECIFICATIONS
C
C ( ENTER NFB OF THE FOLLOWING LINES )
C
C LINE   N   IDIR  VALUE  NEND  NINC
C   N = NODE NUMBER
C   IDIR = DIRECTION OF APPLIED FORCE
C     1 = X
C     2 = Y
C     3 = Z
C
C   VALUE = NUMERICAL VALUE OF FORCE TO BE APPLIED ( LBS )
C   NEND = LAST NODE HAVING THIS APPLIED FORCE. IF OMITTED N IS ASSUMED
C   NINC = INCREMENT TO BE USED FOR FORCE GENERATION FROM N TO NEND.
C     DEFAULT IS 1. ( OPTIONAL )
C
C
      END
```

```
C  X INITIAL NODAL TEMPERATURES
C
C LINE  NTEMPS
C   NTEMPS = NUMBER OF TEMPERATURE INPUT LINES
C
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C   N = NODE NUMBER
C   TEMP = TEMPERATURE ( DEGREES F. )
C   ITYPE = INDICATOR FOR TYPE OF TEMPERATURE REVISION
C     0 = CHANGE NODAL TEMPERATURE TO TEMP
C     1 = INCREMENT NODAL TEMPERATURE BY TEMP
C   NEND = LAST NODE HAVING THE GIVEN TEMPERATURE. IF OMITTED N IS ASSUMED
C   NINC = INCREMENT TO BE USED ON TEMPERATURE GENERATION FROM N TO NEND.
C     IF OMITTED 1 IS ASSUMED
C
C
      END
```

```
C  XI INITIAL ELEMENT PRESSURE LOADS
C
C LINE  NPL
C   NPL = NUMBER OF PRESSURE INPUT LINES
C
C ( ENTER NPL OF THE FOLLOWING LINES )
C
C LINE  NEL  IFACE  P1  P2  P3  P4  P5  P6  P7  P8  NEND  NINC
C   NEL = ELEMENT NUMBER
C   IFACE = FACE NUMBER ( IFACE.GE.1.AND.IFACE.LE.6 )
C   P1 = PRESSURE AT NODE 1 ( PSI )
C   P2, P3, P4, P5, P6, P7, P8 = PRESSURES AT NODES 2,....8 ON THE FACE.
C     IF OMITTED THEY ARE SET EQUAL TO P1. ( POSITIVE PRESSURES INDUCE
C     COMPRESSION IN THE ELEMENT )
C   NEND = LAST ELEMENT HAVING THIS PRESSURE LOADING. IF OMITTED NEL IS
C     ASSUMED
C   NINC = INCREMENT TO BE USED FOR ELEMENT PRESSURE GENERATION FROM
C     NEL TO NEND. IF OMITTED NEL IS ASSUMED
C
C
        END
```

```
C  XII LOAD CASE INFORMATION, FINAL CONDITIONS
C
C     XII.1 LOAD CASE CONTROL CARD
C
C LINE  RPM  IAXIS  IACC  NTI
C
C     XII.2 ACCELERATION SPECIFICATIONS FOR INERTIA OR GRAVITY LOADS
C
C LINE  ACCELX  ACCELY  ACCELZ
C
C
        END
```

```
C  XIII NODAL CONSTRAINED DISPLACEMENTS
C
C LINE  NDB
C
C ( ENTER NDB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C
        END
```

```
C XIV NODAL APPLIED FORCES
C
C LINE  NFB
C
C ( ENTER NFB OF THE FOLLOWING LINES )
C
C LINE  N  IDIR  VALUE  NEND  NINC
C
C
       END
```

```
C XV NODAL TEMPERATURES
C
C LINE NTEMPS
C
C ( ENTER NTEMPS OF THE FOLLOWING LINES )
C
C LINE  N  TEMP  ITYPE  NEND  NINC
C
C
      END
```

Report Documentation Page

National Aeronautics and Space Administration

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
CR - 189089		

4. Title and Subtitle	5. Report Date
3-D Inelastic Analysis Methods for Hot Section Components	January 1992
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
R. L. McKnight, P.C. Chen, L. T. Dame, R. V. Holt, H. Huang, M. Hartle, S. Gellin, D. H. Allen, and W. E. Haisler	
	10. Work Unit No.
	505-63-5B (AERO)
9. Performing Organization Name and Address	11. Contract or Grant No.
General Electric Company Aircraft Engine Business Group Cincinnati, Ohio 45215	NAS 3-23698
	13. Type of Report and Period Covered
12. Sponsoring Agency Name and Address	Final Report
National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191	14. Sponsoring Agency Code

15. Supplementary Notes

Project Manager, Christos, C. Chamis, Structures Division, NASA Lewis Research Center, (216) 433-3252.

16. Abstract

Accomplishments are described for the 2-year program, to develop advanced 3-D inelastic structural stress analysis methods and solution strategies for more accurate and cost effective analysis of combustors, turbine blades, and vanes. The approach was to develop a matrix of formulation elements and constitutive models. Three constitutive models were developed in conjunction with optimized iterating techniques accelerators, and convergence criteria within a framework of dynamic time incrementing. Three formulations models were developed; an eight-noded midsurface shell element, a nine-noded midsurface shell element and a twenty-noded isoparametric solid element. A separate computer program has been developed for each combination of constitutive model-formulation model. Each program provides a functional stand alone capability for performing cyclic nonlinear structural analysis. In addition, the analysis capabilities incorporated into each program can be abstracted in subroutine form for incorporation into other codes or to form new combinations.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Combustors, turbine blades, vanes, matrix formulations, constitutive models, iterating techniques, nine different finite-elements, sample cases, computer codes.	Unclassified-Unlimited Subject Category 39

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified	176	

NASA FORM 1626 OCT 86

PREPARATION OF THE REPORT DOCUMENTATION PAGE

The last page of a report facing the third cover is the Report Documentation Page, RDP. Information presented on this page is used in announcing and cataloging reports as well as preparing the cover and title page. Thus it is important that the information be correct. Instructions for filling in each block of the form are as follows:

Block 1. Report No. NASA report series number, if preassigned.

Block 2. Government Accession No. Leave blank.

Block 3. Recipient's Catalog No. Reserved for use by each report recipient.

Block 4. Title and Subtitle. Typed in caps and lower case with dash or period separating subtitle from title.

Block 5. Report Date. Approximate month and year the report will be published.

Block 6. Performing Organization Code. Leave blank.

Block 7. Author(s). Provide full names exactly as they are to appear on the title page. If applicable, the word editor should follow a name.

Block 8. Performing Organization Report No. NASA installation report control number and, if desired, the non-NASA performing organization report control number.

Block 9. Performing Organization Name and Address. Provide affiliation (NASA program office, NASA installation, or contractor name) of authors.

Block 10. Work Unit No. Provide Research and Technology Objectives and Plans (RTOP) number.

Block 11. Contract or Grant No. Provide when applicable.

Block 12. Sponsoring Agency Name and Address. National Aeronautics and Space Administration, Washington, D.C. 20546-0001. If contractor report, add NASA installation or HQ program office.

Block 13. Type of Report and Period Covered. NASA formal report series; for Contractor Report also list type (interim, final) and period covered when applicable.

Block 14. Sponsoring Agency Code. Leave blank.

Block 15. Supplementary Notes. Information not included elsewhere: affiliation of authors if additional space is required for block 9, notice of work sponsored by another agency, monitor of contract, information about supplements (film, data tapes, etc.), meeting site and date for presented papers, journal to which an article has been submitted, note of a report made from a thesis, appendix by author other than shown in block 7.

Block 16. Abstract. The abstract should be informative rather than descriptive and should state the objectives of the investigation, the methods employed (e.g., simulation, experiment, or remote sensing), the results obtained, and the conclusions reached.

Block 17. Key Words. Identifying words or phrases to be used in cataloging the report.

Block 18. Distribution Statement. Indicate whether report is available to public or not. If not to be controlled, use "Unclassified-Unlimited." If controlled availability is required, list the category approved on the Document Availability Authorization Form (see NHB 2200.2, Form FF427). Also specify subject category (see "Table of Contents" in a current issue of STAR), in which report is to be distributed.

Block 19. Security Classification (of this report). Self-explanatory.

Block 20. Security Classification (of this page). Self-explanatory.

Block 21. No. of Pages. Count front matter pages beginning with iii, text pages including internal blank pages, and the RDP, but not the title page or the back of the title page.

Block 22. Price Code. If block 18 shows "Unclassified-Unlimited," provide the NTIS price code (see "NTIS Price Schedules" in a current issue of STAR) and at the bottom of the form add either "For sale by the National Technical Information Service, Springfield, VA 22161-2171" or "For sale by the Superintendent of Documents, U.S. Government Printing Office, Washington, DC 20402-0001," whichever is appropriate.