NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

Technical Memorandum 33-466

Volume I, Revision 1

VISCEL–A General-Purpose Computer Program for Analysis of Linear Viscoelastic Structures

User's Manual

K. K. Gupta
F. A. Akyuz
E. Heer

JET PROPULSION LABORATORY
CALIFORNIA INSTITUTE OF TECHNOLOGY
PASADENA, CALIFORNIA

October 1, 1972

44 pg.

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION

Technical Memorandum 33-466
Volume I, Revision 1

VISCEL–A General-Purpose Computer Program for Analysis of Linear Viscoelastic Structures

User's Manual

K. K. Gupta
F. A. Akyuz
E. Heer

JET PROPULSION LABORATORY
CALIFORNIA INSTITUTE OF TECHNOLOGY
PASADENA, CALIFORNIA

October 1, 1972

Prepared Under Contract No. NAS 7-100
National Aeronautics and Space Administration

PREFACE

The work described in this report was performed by the Applied Mechanics Division of the Jet Propulsion Laboratory.

ACKNOWLEDGMENT

The authors wish to thank Dr. M. R. Trubert for his continued support and encouragement throughout this program. Thanks are also due to Dr. A. M. Salama, Mr. G. W. Lewis, and Mr. E. N. Duran for their assistance in the development of this work.

Editorial assistance of Mr. Harold M. Yamamoto is acknowledged with sincere appreciation.

CONTENTS

TABLES

FIGURES

CONTENTS (contd)

ABSTRACT

This revised user's manual describes the details of a general-purpose computer program VISCEL (VISCoELastic analysis) which has been developed for the analysis of equilibrium problems of linear thermoviscoelastic structures. The program, an extension of the linear equilibrium problem solver ELAS, is an updated and extended version of its earlier form (written in FORTRAN II for the IBM 7094 computer). A synchronized material property concept utilizing incremental time steps and the finite element matrix displacement approach has been adopted for the current analysis. Resulting recursive equations incorporating memory of material properties are solved at the end of each time step of the general step-by-step procedure in the time domain. A special option enables employment of constant time steps in the logarithmic scale, thereby reducing computational efforts resulting from accumulative material memory effects. A wide variety of structures with elastic or viscoelastic material properties can be analyzed by VISCEL.

The program is written in FORTRAN V language for the UNIVAC 1108 computer operating under the EXEC 8 system. Dynamic storage allocation is automatically effected by the program, and the user may request up to 195K core memory in a 260K UNIVAC 1108/EXEC 8 machine. The physical program VISCEL, consisting of about 7200 instructions, has four distinct links (segments), and the compiled program occupies a maximum of about 11700 words decimal of core storage. VISCEL is stored on magnetic tape, and is available from the Computer Software Management and Information Center (COSMIC).

VISCEL – A GENERAL-PURPOSE COMPUTER PROGRAM FOR ANALYSIS OF LINEAR VISCOELASTIC STRUCTURES

USER'S MANUAL

I. INTRODUCTION

The general-purpose digital computer program VISCEL is capable of solving equilibrium problems associated with one-, two-, or three-dimensional linear viscoelastic structures (Fig. A-1). Since the program is an extension of the linear equilibrium problem solver ELAS (Ref. 1), its solution at the beginning of the initial time step yields elastic solution of structures. Basic inputs of VISCEL, thus, are the same as in ELAS; additional inputs are, however, necessary for VISCEL, which represent changes in material properties and loading in the time domain. Other important features of the program include dynamic memory allocation, optional node relabelling scheme, boundary condition imposition during assembly of the stiffness matrix and its storage within a variable bandwidth. The program is further divided into four distinct links, namely, input, generation, deflection, and stress links.

This user's manual describes the numerical problem formulation, input preparation, output description, and other relevant details of the program. The physical program is available from COSMIC.[1]

Volume II of this report is the program manual, which contains the lists of variables, subroutines, and flow charts as well as other pertinent program information (Ref. 2).

---

[1] Computer Software Management and Information Center, Computer Center, University of Georgia, Athens, Georgia 30601, telephone: (404) 452-3265.

II. BASIC CAPABILITIES OF THE VISCEL PROGRAM

The basic capabilities and the initial inputs of VISCEL are the same as the linear equilibrium problem solver ELAS (Ref. 1). In order to achieve a self-contained report, this report includes several tables and figures from Ref. 1; they are provided in the Appendix in appropriate order. Thus, Tables A-1 and A-2 describe, respectively, the various structures that can be solved by VISCEL and their compatible combinations. Also, information regarding various available finite elements is given in Tables A-3, A-4, and A-5. Further, the usual conventions for ordering of element nodes are explained in Table A-6. VISCEL can handle any material, namely, isotropic, orthotropic, or anisotropic; their input requirements are described in Fig. A-2.

III. NUMERICAL FORMULATION OF THE LINEAR THERMOVISCOELASTIC PROBLEM

Reference 3 gives complete derivation of the numerical formulations of the linear thermoviscoelastic problem, whereas Ref. 4 presents details of the finite element technique. However, such formulations are summarized below in a simplified manner for completeness of this report.

A. Basic Approach

The fundamental equilibrium problem in structural analysis can be formulated as differential equations with appropriate boundary conditions; alternatively, an equivalent extremum formulation may be developed based on the principle of minimum potential energy and its complement (Ref. 5). In this work, structural discretization is achieved by the finite element displacement matrix technique, a variant of the well-known Ritz method for the minimization of the total potential energy functional ψ associated with admissible displacement trial functions. The admissible functions are restricted to be sufficiently smooth, usually being algebraic or trigonometric polynomials, and, furthermore, they are required to satisfy essential boundary conditions arising from the requirement of geometric compatibility. This is achieved by expressing the trial solution in terms of a set of linearly independent known functions and undetermined parameters and then minimizing the functional with respect to such parameters.

In the finite element method, a structure is discretized by any suitable random mesh, and a family of piecewise continuous displacement fields is prescribed for each element, which are finally expressed in terms of their nodal function values. Such nodal displacements are the undetermined parameters to be determined from the extremum principle, the fundamental assumption in the procedure being that the total potential energy of the entire structure is equal to the sum of potential energies of the individual elements (Ref. 4). Such an assumption is valid provided the displacement functions and their derivatives of order one less than the highest one appearing in the functional are continuous at interelement boundaries; this ensures that values of highest derivatives occurring in the total potential energy functional ψ remains finite (Ref. 4). Obviously, the greater the number of chosen undetermined parameters, i.e., finer the finite element mesh, the lower the value of the total potential energy would be, yielding even better approximations. Whereas ψ approaches its minimum value from above, the corresponding strain energy value is always underestimated, and hence the present approach computes lower bounds of associated displacements. The finite element procedure thus gives a stationary value of ψ for the variations of the unknown nodal displacements. Because of its resemblance to the piecewise Ritz procedure, any particular nodal parameter is only influenced by its adjacent elements, and hence the final stiffness matrix is highly banded in nature for most practical problems. It can further be shown that the minimization process of total potential energy of the entire structure with respect to each unknown nodal displacement is equivalent to the appropriate summation of such process for all individual elements with respect to their nodal parameters. For quadratic functionals, the piecewise Ritz procedure for each element yields symmetric linear equations in the element displacement vector. The minimization process for the entire structure then leads to the set of linear, simultaneous equations:

$$\frac{\partial \psi}{\partial \mathbf{q}} = \mathbf{K}\mathbf{q} + \mathbf{P} = \sum \mathbf{k}^e \mathbf{q}^e + \sum \mathbf{P}^e = 0 \qquad (1)$$

where

$\mathbf{k}^e$ = element stiffness matrix

$\mathbf{q}^e$ = element nodal displacement vector

$\mathbf{p}^e$ = equivalent nodal load vector

with appropriate summation over all elements based on nodal connectivity. Such equations are further positive definite for stable structures and may be solved by standard processes to yield the undetermined nodal displacements. Computation of stresses, etc., is performed next by the usual procedure. The program starts with the computation of element stiffness matrices already derived above.

In viscoelasticity, the creep strain rate or the relaxation stress response is dependent not only on the current stress and strain state, but also on the entire history of its development in the time domain. Associated numerical computation procedures usually adopt a step-by-step incremental process, which normally requires knowledge of stress and strain at all preceding intervals. This enables computation of stresses/strains at a given time implied by some relevant law of the characteristic functions. Usually such material properties are strongly dependent on time and temperature. The viscoelastic equations are developed as finite difference equations in time and finite element matrix equations in space (Ref. 3). This computer program is based on linear thermoviscoelastic formulations utilizing a "synchronized" material property concept for thermorheologically simple materials. The fundamental assumptions may be summarized as follows:

(1) Material properties. The material properties may be temperature-dependent and are assumed to behave in a thermorheologically simple way; thus, for temperature changes, the characteristic functions, both creep and relaxation, show pure shift when they are plotted against the logarithm of time.

Such materials are better suited for a complete characterization over a large range of time and temperature scale since their rheological behavior can be described for the entire temperature range as a single function of reduced time and temperature.

Thus, when any characteristic function, such as the relaxation modulus, is plotted against reduced time, all curves will fall on the single curve for initial temperature T_0. Hence it is then necessary to determine relaxation/creep functions for one temperature only.

The shift functions may sometimes be dependent on stresses, requiring determination of the shift function at the end of each time step. However, such considerations are excluded in the present version of the program. The material can be isotropic, orthotropic, or general (Fig. A-2), provided they are properly defined by experimental results. For this analysis, it is required to have a knowledge of the modulus functions (relaxation-type functions). Furthermore, the material is assumed to be at least slightly compressible.

The concept of synchronized material properties is that all material properties are functions of only one parameter ξ. The same concept applies to external loadings, both mechanical and/or thermal. The parameter ξ may be time, reduced time, or any other suitable variable. Material and load data are considered in functional form (Fig. 1), which are to be presented at each time step, in the shape of predetermined tabulated values obtained either experimentally or derived from analytical considerations; any interaction between them is assumed to be included in such values.

(2) Linear viscoelastic behavior. Strains are linear functions of stresses, but are strongly dependent on loading history, implying that if all loads are doubled, all deformations will be doubled too. Thus, creep/relaxation laws are linear in stress/strain and as such the principle of superposition is valid for such cases. Further geometric nonlinearies, e.g., large strain or large deformations, are not considered for the current analysis.

(3) Deflection boundary conditions. Deflection boundary conditions are assumed to remain unaltered throughout the entire time domain of computation, being fixed initially at the beginning of

the initial time step. The solution at the beginning of such initial time step corresponds to the usual linear elastic analysis of the structure.

B. Analysis Review

Numerical formulation of the step-by-step linear thermoviscoelastic analysis procedure for quasi-static problems may now be summarized. A basic assumption in the analysis is that the materials are thermorheologically simple in nature. Such an assumption is necessary so that the characteristic functions may be singly defined for the entire temperature range in the time domain. The usual field equations for viscoelastic materials may then be extended for the thermoviscoelastic case. This is achieved by introducing the concept of a "reduced time" when all characteristic functions fulfill the same time-temperature shift and can be represented as a function of reduced time:

$$\xi(x_h, t) = \int_0^t \frac{d\tau}{a[T(x_h, \tau)]} \tag{2}$$

in which a(T) is the time shift function usually determined experimentally as a function of temperature T only. Such shift function dependence on time t and position x_h within the material region is implicit through T, and may be sometimes described by the well-known Williams-Landel-Ferry (WLF) equation. Relationship (2) signifies that all the characteristic functions, such as relaxation moduli of a thermoviscoelastic material at any arbitrary temperature T corresponding to time t, may now be expressed by their behavior at reference temperature T_0 on the new reduced time scale ξ. Each relaxation modulus, signifying relaxation stress variation for unit strain applied initially, may then be expressed as

$$E^{T}_{ijk\ell}(t) = E^{T_0}_{ijk\ell}(\xi) \quad (i = j = k = \ell = 1, 2, 3) \tag{3}$$

$E_{ijk\ell}(t)$ being the general anisotropic relaxation moduli having 21 independent components. The constitutive equations for the usual viscoelastic case may be derived by approximating strain variations by the sum of a series of step functions, which corresponds to a series of relaxation displacement inputs. Constitutive equations are obtained, from superposition principles, in the form of hereditary integrals. For the present thermoviscoelastic case, the constitutive equations may simply be derived from such relations for the corresponding viscoelastic case by utilizing Eq. (3):

$$\sigma_{ij}(x_h, t) = \int_{-\infty}^{t} E_{ijk\ell}\left[\xi(x_h, t) - \xi'(x_h, \tau)\right]$$

$$\times \frac{\partial}{\partial \tau}\left[e_{k\ell}(x_h, \tau) - \alpha_k \; (x_h, \tau)\theta(x_h, \tau)\right] d\tau \qquad (4)$$

which may be rewritten as

$$\sigma_{ij} = E_{ijk\ell}(\xi)\, e_{k\ell(0)} + \int_{0}^{t} E_{ijk\ell}(\xi - \xi')\frac{\partial}{\partial \tau}(e_{k\ell} - \alpha_{k\ell}\theta)\, d\tau \qquad (5)$$

in which $e_{k\ell(0)}$ is the initially induced step strain at $t = 0$, the corresponding first term being the effect of such initial strain at time $\xi(x_h, t)$. The kernel of the hereditary integral $E_{ijk\ell}[\xi(x_h, t) - \xi(x_h, \tau)]$ may be considered as the memory function transforming the influence of pulse strain at time τ to the time instant t. In addition to Eq. (4), two more equations are required to completely define the field equations:

(1) Equilibrium equations

$$\sigma_{ij,j} + f_i = 0 \qquad (6)$$

(2) Strain-displacement equations

$$e_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i}) \qquad (7)$$

where f_i is the body force component per unit volume. The field equations may next be expressed as incremental field equations when the time domain is subdivided into arbitrary intervals $\Delta t(m)$. Equations (6) and (7) and the stresses of Eq. (5) then take the following form:

$$\left.\begin{aligned}\Delta\sigma_{ij(m),j} + \Delta f_{i(m)} &= 0\\ \Delta e_{ij(m)} &= \frac{1}{2}(\Delta u_{i(m),j} + \Delta u_{j(m),i})\end{aligned}\right\} \qquad (8)$$

$$\sigma_{ij(n)} = E_{ijk\ell}(\xi_{(n)})\, e_{k\ell(0)} + \int_0^{t_{(n)}} E_{ijk\ell}(\xi_{(n)} - \xi')\frac{\partial}{\partial\tau}(e_{k\ell} - \alpha_{k\ell}\theta)\, d\tau \qquad (9)$$

Finally, Eq. (9) may be approximated and expressed in the matrix form as follows:

$$\left\{\sigma_n\right\} = \left[\mathbf{E}(\xi_n)\right]\left\{e_0\right\} + \sum_{m=1}^{m=n}\left[\mathbf{E}(\xi_n - \xi_{n-1})\right]\left\{\Delta \mathbf{e}_m - \Delta(\alpha\theta)_m\right\} \qquad (10)$$

The continuum is next divided into small finite elements, and piecewise continuous displacement fields are prescribed for each of such elements in terms of their time-dependent nodal function values. Minimization of the total potential energy with respect to such parameters then yields the incremental equilibrium load-deflection equations of the entire structure. Such step-by-step incremental equations may finally be written in the global coordinate system:

$$\left[\mathbf{K}_{n,n-1}\right]\left\{\Delta\mathbf{U}_n\right\} = \left\{\mathbf{P}_n\right\} - \sum_{m=1}^{m=n-1}\left[\mathbf{K}_{n,m-1}\right]\left\{\Delta\mathbf{U}_m\right\} - \left[\mathbf{K}_n\right]\left\{\mathbf{U}_0\right\}$$

$$+\sum_{m=1}^{n}\left\{\mathbf{T}_{n,m-1}\right\} + \left\{\mathbf{F}_n\right\} \qquad (11)$$

with

$\left[K_{i,j}\right]$ = stiffness matrix derived from material matrix computed for the reduced time difference $\Delta\xi_{ij} = \xi_j - \xi_i$

$\left\{P_n\right\}$ = external load vector at step n

$\left\{T_{n,m-1}\right\}$ = forces due to temperature changes

$\left\{F_n\right\}$ = body forces vector

and in which the summation, as usual, signifies the memory of the material. The element stresses may then be obtained from Eq. (10), when element strains are derived from the usual relationship:

$$\bar{\mathbf{u}}^e = \lambda \mathbf{U}^e$$

$$\mathbf{u} = \mathbf{a}\bar{\mathbf{u}}^e$$

$$\mathbf{e} = \mathbf{b}\mathbf{u}$$

$\mathbf{U}^e$, $\bar{\mathbf{u}}^e$ being element nodal displacements in the global and local coordinate systems, respectively, λ the direction cosine matrix, and $\mathbf{u}$ and $\mathbf{e}$ are displacements and strains within the element.

C. Special Incremental Procedure

It is apparent from the nature of Eq. (11) that computation time may be excessive after a few time steps. This is because at each time step, recomputation of solution results is required for all preceding time steps, which are then added to obtain the final solution. However, in order to minimize such computation efforts, the program provides an option by which time steps may be so chosen that previous time intervals become a subset of the following time intervals. Thus, the parameter ξ may be expressed as the summation of incremental $\Delta\xi$'s as follows (Ref. 2):

$$\xi_j^i = \sum_{i=1}^{M} \sum_{j=1}^{N(i)} \left(N(i)^{i-1} \Delta\xi \right)_j \qquad (12)$$

where M defines the total number of time step groups, and N(i) is the number of steps in the ith group. The values of M and N can be suitably chosen by the user, and this scheme may be employed to solve the recursive Eqs. (10) and (11), provided material property and external loads are available for each ξ_j^i. Figure 2 shows details of such a computation scheme for values of M = 3, N(1) = 2, N(2) = 3, and N(3) = 2. In such cases the time intervals tend to remain constant in the logarithmic scale; thus, it is then possible to cover a long time domain with relatively small computational effort.

IV. INPUT PREPARATION FOR VISCEL

The program VISCEL is assumed to be stored on a tape (say, number 12345), which contains the symbolic and relocatable program elements. Data deck corresponding to any problem must be preceded by a set of control cards which are first described below. The actual data deck preparation is explained next.

A. VISCEL Control Cards for UNIVAC 1108/EXEC 8 Computer

Depending on the size of the problem to be solved, the user may request for an appropriate core storage. Such values are assigned to an integer LDATA, to be calculated approximately from Ref. 2, Fig. 1, in which for most problems the major storage space would be required for elements of the upper symmetric half of the stiffness matrix. The program is compiled for LDATA = 20000 words decimal storage, and if more storage is requested, this is achieved by recompilation of two small programs COMBK and MAIN, the block data and the main driver programs, respectively.

Furthermore, as explained in Ref. 2 (pp. 3-4), various Fastrand (drum) file storage units are utilized as additional stores during execution of the program; their functions are summarized in Table A-7. Unless specified, the UNIVAC 1108 system automatically allocates 128 tracks to each of the units, which, however, may be inadequate for solution of large order problems. It is then necessary to increase the number of data tracks for such units by inserting relevant control cards in the run stream.

Control cards corresponding to the two sets of values of LDATA are as follows:

(1) Control cards with LDATA ≤ 20000

The following run stream may be used for problems which do not require more than 20000 words storage for the COMMON:

```
@RUN,/TPC   RUNID, ACCOUNT, PROJECT, TIME, PAGES
@MSG,   READ TAPE   12345
@ASG, T   TAPE, T, 12345R
@FREE   TPF$
@ASG, T   TPF$, F///500
@COPY, G   TAPE, TPF$
@FREE   TAPE
@ASG, T   UNIT NUMBER, F2///1000
@XQT   ABSEL
VISCEL INPUT CARDS
@FIN
```

(2) Card input with LDATA > 20000 (say, LDATA = 80000)

When COMMON requirements are greater than 20000 (say, 80000), the following typical run stream may be adopted:

```
@RUN, /TPC   RUNID, ACCOUNT, PROJECT, TIME, PAGES
@MSG,   READ TAPE 12345
@ASG, T   TAPE, T, 12345R
@FREE   TPF$
@ASG, T   TPF$, F///500
```

```
@COPY, G   TAPE, TPF$
@FREE TAPE
@ASG, T   UNIT NUMBER, F2///1000
@FOR, S   COMBK, COMBK, COMBK
 -2, 2
          PARAMETER LDATA = 80000
@FOR, S   MAIN, MAIN, MAIN
 -2, 2
          PARAMETER LDATA = 80000
@PACK
@PREP
@MAP, EN   MAPEL, ABSEL
@XQT   ABSEL
VISCEL INPUT CARDS
@FIN
```

Requests for additional storage tracks for the Fastrand units may be made by inserting control cards, shown above within the dotted boundaries.

B. Input of Problem Data

The physical arrangement of the data deck which follows the control cards (explained in previous section) is depicted in Fig. 3. This deck corresponds to values M = 2, N(1) = 2, N(2) = 3 in the time domain defined by Eq. (12). VISCEL input data may be as described below, with reference to Table 1 describing input items; the integers of the problem control card (Table 1, input item 2) is explained in Table A-8.

Data Group 1: Basic input for the elastic problem which also corresponds to the initial time solution of the viscoelastic problem

Data Group 2: Data for multiple solutions of the elastic problem

or

Data for viscoelastic incremental solution in the time domain

The nature of the data in group 2, if any, is determined by the contents (ISUCA value) of the END card in the master (initial time) deck (input item 19, Table 1) and the subsequent additional input data decks for viscoelastic problems. Field specification for the END card is as follows:

$$70X, I7, 3HEND \tag{13}$$

in which the I7 field corresponds to the integer ISUCA, which is to be set as follows:

(1) ISUCA $= 0$ For linear elastic problems

(2) ISUCA < 0 For multiple runs

(3) ISUCA > 0 For linear viscoelastic problems

(4) ISUCA $= 1$ For master and following deck

(5) ISUCA > 1 For following additional decks in increasing sequence

Thus, in the viscoelastic case, the first card following the data of the previous time step is the problem control card, equivalent to input item 2 of the initial time step, containing information on modifiable input items. The modified information is provided next, followed by the END card with the ISUCA value which determines the nature of the data, if any, in the succeeding step. A numerical example of a two-dimensional plane stress problem (Fig. 4) with irregular mesh labeling is chosen to elaborate on the preparation of the data; the complete input data are presented in Fig. 5 with $M = 2$, $N(1) = 4$, and $N(2) = 2$ values selected for the incremental time scheme of Eq. (12). Node relabeling may be requested by using appropriate option in input item 17 of Table 1.

Relevant details on permanent and modifiable input items are provided in Table 2. Element data corresponding to input item 16 of Table 1 is described in Table A-9. Also, input items 13, 15, and 18 in the same table may be specifically described as follows:

(1) Input item 13 (angle types - fixing local y and z axes)

In connection with element type 4 (Table A-3), the input corresponding to column 16 of Table A-4 consists of a list of ϕ angles in degree units. The ϕ values are assigned quantities with absolute values less than 90 deg and are defined as the angle between the local y and global Y axes. Let the direction cosine vectors be denoted by $(\ell_{xX}, \ell_{xY}, \ell_{xZ})$, $(\ell_{yX}, \ell_{yY}, \ell_{yZ})$ and $(\ell_{zX}, \ell_{zY}, \ell_{zZ})$ in which the local x axis is assumed to coincide with the nodal line 1-2 (Table A-3). Then the signs of ℓ_{xX}, ℓ_{yX}, and ℓ_{zY} are used to determine the sign of ϕ; such procedure is summarized in Table A-10.

(2) Input item 15 (deflection boundary conditions)

The deflection boundary condition relations may be written as (Ref. 1):

$$u_{i,j} = a_0 + a_1 u_{i',j'} + a_2 u_{i'',j''} + \ldots \tag{14}$$

when coefficients $a_0, a_1, a_2, \ldots$, and the input pairs (i, j), (i', j'), (i'', j''), . . . , are the relevant inputs as follows:

i, j	i, j	a_0
i, j	i', j'	a_1
i, j	i'', j''	a_2
.	.	.
.	.	.
.	.	.

in which the first two pairs along each row are the two degrees of freedom, under consideration and the related one, the third scaler relating such two deflection components.

(3) Input item 18 (concentrated load input)

The inputs for the prescribed force boundary conditions for concentrated loads are as follows:

i, j	P
i', j'	P'
.	.
.	.
.	.

where $P_{i,j}$, $P'_{i',j'}$, ..., are the prescribed concentrated nodal loads at nodes i, i', ..., corresponding to degrees of freedom j, j', ..., respectively. Apart from concentrated loads, the elements may be subjected to any pressure as well as temperature loading as indicated in Table A-4.

V. DESCRIPTION OF VISCEL OUTPUT

Table A-11 provides a list of output items of the initial time step solution, whereas Table 3 summarizes such items for the entire viscoelastic problem with an input index value INP set to 1 for the elastic solution. The definition of stress components at mesh points is given in Table A-12.

VI. ERROR MESSAGES AND DIAGNOSTICS

The error messages shown in Table A-13 are usually related to the initial time step solution. Error message 10 in particular needs a detailed explanation, which appears either for geometrically unstable structures, or when the structure is not adequately supported. The last number appearing in the error message, if negative, indicates the mesh number to be checked carefully for existence of any unknown deformation. However, if the number is positive, then it is first necessary to find from output item 10 of Table A-11 the pair of numbers with the second number identical to this error message number. The first number of the pair is called IBB, denoting the equation number in the reduced set of the stiffness matrix. Then column IBB of output item 10 of Table A-11 is searched for the row having the same IBB number found previously, such that the column IBO contains the number -1. The mesh number in that row happens to be the trouble spot, whereas the defective direction is the one appearing in the table heading of the output item. In such case, the element descriptions, material matrices, and geometric continuity around the mesh point are to be checked to correct the situation.

VII. CONCLUDING REMARKS

This user's manual describes in detail the information necessary to utilize the computer program VISCEL for the solution of thermoviscoelastic problems associated with practical structures. Extensive applications of the problem are envisaged in the analysis of a wide variety of practical structures including solid propellant rocket motors, spacecraft components such as solar panels, etc. In order to make this document complete, some information, including most tables and figures in the Appendix, have been reproduced from Ref. 1.

REFERENCES

1. Utku, S., and Akyuz, F.A., ELAS – A General-Purpose Computer Program for the Equilibrium Problems of Linear Structures: Vol. I. User's Manual, Technical Report 32-1240. Jet Propulsion Laboratory, Pasadena, Calif., Feb. 1, 1968.

2. Gupta, K.K., and Akyuz, F.A., VISCEL – A General-Purpose Computer Program for Analysis of Linear Viscoelastic Structures: Vol. II. Program Manual, Technical Memorandum 33-466. Jet Propulsion Laboratory, Pasadena, Calif., July 15, 1972.

3. Heer, E., and Chen, J.C., Finite Element Formulation for Linear Thermoviscoelastic Materials, Technical Report 32-1381. Jet Propulsion Laboratory, Pasadena, Calif., June 1, 1969.

4. Zienkiewicz, O.C., The Finite Element Method in Engineering Science. McGraw-Hill Book Co., Inc., New York, 1971.

5. Crandall, S.H., Engineering Analysis. McGraw-Hill Book Co., Inc., New York, 1956.

Table 1. Input items (summary of options, contents, and formats)

Input item No.	Conditions determining options	List of input statements that read the associated input item card(s)[a]	Format (outside parentheses indicate the possibility of multiple cards)
1		(B_i, $i = 1, 14$) The card may contain any alphanumeric message	14A6
2		IN, IT, IDEG, ITYPE, IGEM, ISTR, IH, I8, IBN, IP, IPRS, IMAT, NTIC, ISDT, ISDY, ISDZ, IARE, IMMX, IMMY, IMMZ, IMFI, INX, INP, ISHUF, ICOR, IBUN, IMES, IPIR, ITAP, ITAS, G1, G2, G3, ACEL	2I4, 6I1, 3I4, 10I2, 9I1, 3F5.4, E10.5 (see Table A-8 for details)
3[b]	ITYPE = 0	(i, E_i, G_i, α_i, $i = 1$, IMAT)	(3(I2, 3E8.5))
	ITYPE = 1	(i, D'_{11_i}, D'_{12_i}, D'_{14_i}, D'_{22_i}, D'_{24_i}, D'_{44_i}, D'_{55_i} D'_{56_i}, D'_{66_i}, i, α'_{1_i}, α'_{2_i}, $i = 1$, IMAT)	(I2, 9E8.5/I2, 2E8.5)
	ITYPE = 2	(i, D_{11_i}, D_{12_i}, D_{13_i}, D_{14_i}, D_{15_i}, D_{16_i}, D_{22_i}, D_{23_i}, D_{24_i}, i, D_{25_i}, D_{26_i}, D_{33_i}, D_{34_i}, D_{35_i}, D_{36_i}, D_{44_i}, D_{45_i}, D_{46_i}, i, D_{55_i}, D_{56_i}, D_{66_i}, α_{1_i}, α_{2_i}, α_{3_i}, $i = 1$, IMAT)	(I2, 9E8.5/I2, 9E8.5/I2, 6E8.5)
4	$1 \leq$ IPRS ≤ 99	(i, p_i, $i = 1$, IPRS)	(8(I2, E8.5))
	IPRS = 0	No input card	
5	$1 \leq$ NTIC ≤ 99	(i, h_i, $i = 1$, NTIC)	(8(I2, E8.5))
	NTIC = 0	No input card	
6	$1 \leq$ ISDT ≤ 99	(i, Δt_i, $i = 1$, ISDT)	(8(I2, E8.5))
	ISDT = 0	No input card	
7	$1 \leq$ ISDY ≤ 99	(i, $(\partial t/\partial y)_i$, $i = 1$, ISDY)	(8(I2, E8.5))
	ISDY = 0	No input card	
8	$1 \leq$ ISDZ ≤ 99	(i, $(\partial t/\partial z)_i$, $i = 1$, ISDZ)	(8(I2, E8.5))
	ISDZ = 0	No input card	
9	$1 \leq$ IARE ≤ 99	(i, A_i, $i = 1$, IARE)	(8(I2, E8.5))
	IARE = 0	No input card	
10	$1 \leq$ IMMX ≤ 99	(i, C_i, $i = 1$, IMMX)	(8(I2, E8.5))
	IMMX = 0	No input card	
11	$1 \leq$ IMMY ≤ 99	(i, I_{y_i}, $i = 1$, IMMY)	(8(I2, E8.5))
	IMMY = 0	No input card	
12	$1 \leq$ IMMZ ≤ 99	(i, I_{z_i} $i = 1$, IMMZ)	(8(I2, E8.5))
	IMMZ = 0	No input card	
13	$1 \leq$ IMFI ≤ 99	(i, ϕ_i, $i = 1$, IMFI)	(8(I2, E8.5))
	IMFI = 0	No input card	
14	ICOR = 0 $2 \leq$ IN ≤ 9999	(j, X_j, Y_j, Z_j, $j = 1$, IN)	(I4, 3E12.4, 40X) or (40X, I4, 3E12.4) or (2(I4, 3E12.4))
	ICOR = 1	Input card(s) should be as required by the user's version of subroutine CORG (see Ref. 1)	

Table 1 (contd)

Input item No.	Conditions determining options	List of input statements that read the associated input item card(s)[a]	Format (outside parentheses indicate the possibility of multiple cards)
15	IBUN = 0 $1 \leq$ IBN ≤ 9999	$(i_k, j_k, i'_k, j'_k, a_k, k = 1, \text{IBN})$	(5(I4,I1, I4, I1, F6.0))
	IBUN = 1	Input card(s) should be as required by the user's version of subroutine BUNG (see Ref. 1)	
16	IMES = 0 $1 \leq$ IT ≤ 9999	$(\text{MM}_m, \text{J1W}_m, \text{J2W}_m, \text{J3W}_m, \text{J4W}_m, \text{J5W}_m, \ldots, m = 1, \text{IT})$	(20I4) (see Table A-9 for variables of the list)
	IMES = 1	Input card(s) should be as required by the user's version of subroutine MESG (see Ref. 1)	
17	ISHUF = 0 or 1	No input card	
	ISHUF = 2	$(N_i, i = 1, \text{IN})$	(20I4)
	ISHUF = 3	$(N_i, IMAX_i, i = 1, \text{IN})$	(20I4)
18	$1 \leq$ IP ≤ 9999	$(i_l, j_l, P_l, l = 1, \text{IP})$	(5(I4, I1, E11.4))
	IP = 0	No input card	
19		No list (the card is punched END in the last three columns)	70X, I7, 3HEND (ISUCA value)
20		No input for standard VISCEL; otherwise input of certain user's subroutines (see Ref. 1)	
VISCEL PROBLEM CONTROL CARD, PROVIDES MODIFIABLE INFORMATION			
MODIFIED INFORMATIONS | | |			
19		END card	70X, I7, 3HEND (ISUCA value)
PROCESS TO BE REPEATED FOR SUBSEQUENT TIME STEPS			

[a]Nomenclature

p	pressure
h	thickness
Δt	temperature increase
$\partial t/\partial y$	temperature gradient in local y-axis direction
$\partial t/\partial z$	temperature gradient in local z-axis direction
A	cross-sectional area
C	torsional constant
I_y	moment of inertia about local y axis
I_z	moment of inertia about local z axis
ϕ	angle determining the orientation of principal axes of cross section in overall coordinate system
X, Y, Z	overall coordinates of mesh points
x, y, z	local coordinates
$(i_k, j_k), (i'_k, j'_k), a_k$	index pairs and the constant of the kth dbc input unit (see Section III-D)
i_l, j_l, P_l	index pair and constant of the lth concentrated load input unit (see Section IV-B)

[b]The symbols shown in Input Item 3 are defined in Figs. A-2c, 2d, and 2e.

Table 2. Permanent and modifiable input items

<table>
<tr><th>Input item number</th><th>Description of input item</th><th>Qualifications</th><th>Existence in the master deck</th><th>Existence in the successive decks</th></tr>
<tr><td>1</td><td>Title card</td><td>For master deck only</td><td>*</td><td>-</td></tr>
<tr><td>2</td><td>Control card</td><td>For master and successive decks</td><td>*</td><td>*</td></tr>
<tr><td colspan="5">Modifiable information</td></tr>
<tr><td>3</td><td>Material types</td><td>For master and successive decks</td><td>*</td><td>o</td></tr>
<tr><td>4</td><td>Pressure types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>5</td><td>Thickness types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>6</td><td>Temperature increase types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>7</td><td>Temperature gradient types -local y-axis direction</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>8</td><td>Temperature gradient types -local z-axis direction</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>9</td><td>Cross-sectional area types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>10</td><td>Torsional constant types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>11</td><td>y-moment-of-inertia types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>12</td><td>z-moment-of-inertia types</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>13</td><td>Angle types-fixing local y and z axes</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td colspan="5">Permanent information</td></tr>
<tr><td>14</td><td>Mesh point coordinates</td><td>For master deck only</td><td>o</td><td>-</td></tr>
<tr><td>15</td><td>Deflection boundary conditions</td><td>For master deck only</td><td>o</td><td>-</td></tr>
<tr><td>16</td><td>Element descriptions</td><td>For master deck only</td><td>o</td><td>-</td></tr>
<tr><td>17</td><td>Relabelling information</td><td>For master deck only</td><td>o</td><td>-</td></tr>
<tr><td colspan="5">Modifiable information</td></tr>
<tr><td>18</td><td>Concentrated loads</td><td>For master and successive decks</td><td>o</td><td>o</td></tr>
<tr><td>19</td><td>End card</td><td>For master and successive decks</td><td>*</td><td>*</td></tr>
</table>

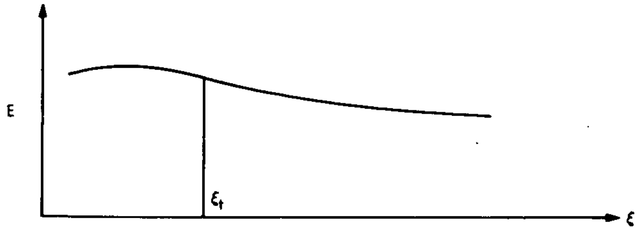
* = the card(s) must exist.

o = the card(s) exist optionally depending on the control constant in the control card and relate to the modifiable information.

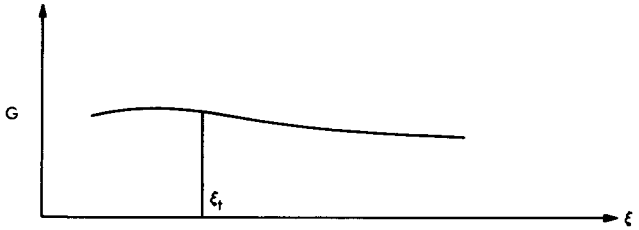
\- = the card(s) must not exist.

Table 3. Summary of output items

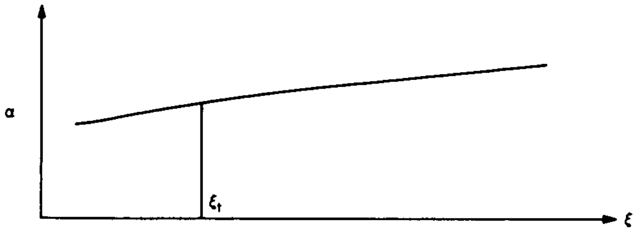
Output item number	Output for linear elastic solution or initial time solution of linear viscoelastic problems	Output for linear viscoelastic solution
1	Linear elastic problem or linear viscoelastic problem (1) Title of the problem (2) Table for control constants	Number of equal time step group. . . . Number of time steps in the group. . . .
2	Modifiable information (material properties, pressure types, etc.)	Modifiable information (material properties, pressure types, etc.)
3	Nodal coordinates	
4	Mesh topology; element property types	
5	Relabelling message	
6	Topology of the reduced stiffness matrix	
7	Stiffness matrix requires. . . . storage locations	
8	Total common length is (decimal). . . . storage locations	
9	Count of main diagonal elements of row listed stiffness matrix	
10	Force and displacement boundary conditions in directions 1 (2, 3, 4, 5, 6)	
13	Input link took. . . . seconds	Input link took. . . . seconds
17	Generation link took. . . . seconds	Generation link took. . . . seconds
19	Nodal deflections	Accumulative nodal deflections
20	Forces acting at the nodes	Forces acting at the nodes
21	Deflection link took. . . . seconds	Deflection link took. . . . seconds
22	Stresses at the nodes	Stresses at the nodes
25	Stress link took. . . . seconds	Stress link took. . . . seconds



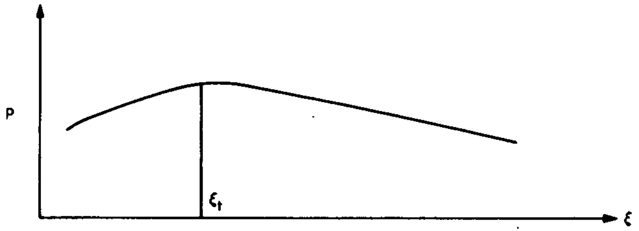
(a) ELASTIC MODULUS



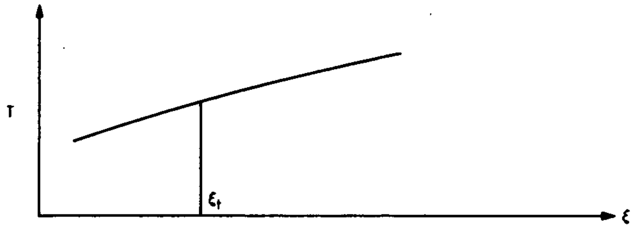
(b) SHEAR MODULUS



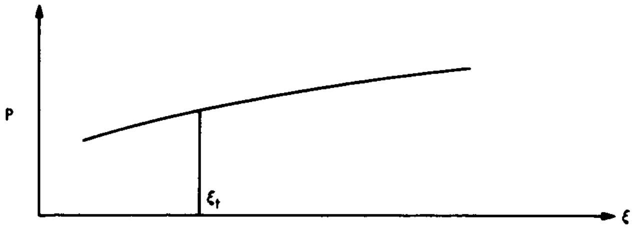
(c) COEFFICIENT OF EXPANSION



(d) EXTERNAL PRESSURE



(e) TEMPERATURE CHANGE



(f) OTHER PROPERTIES (MODIFIABLE), CONCENTRATED LOAD, ETC.

Fig. 1. Schematic representation of the material properties and external disturbances at ξ_t

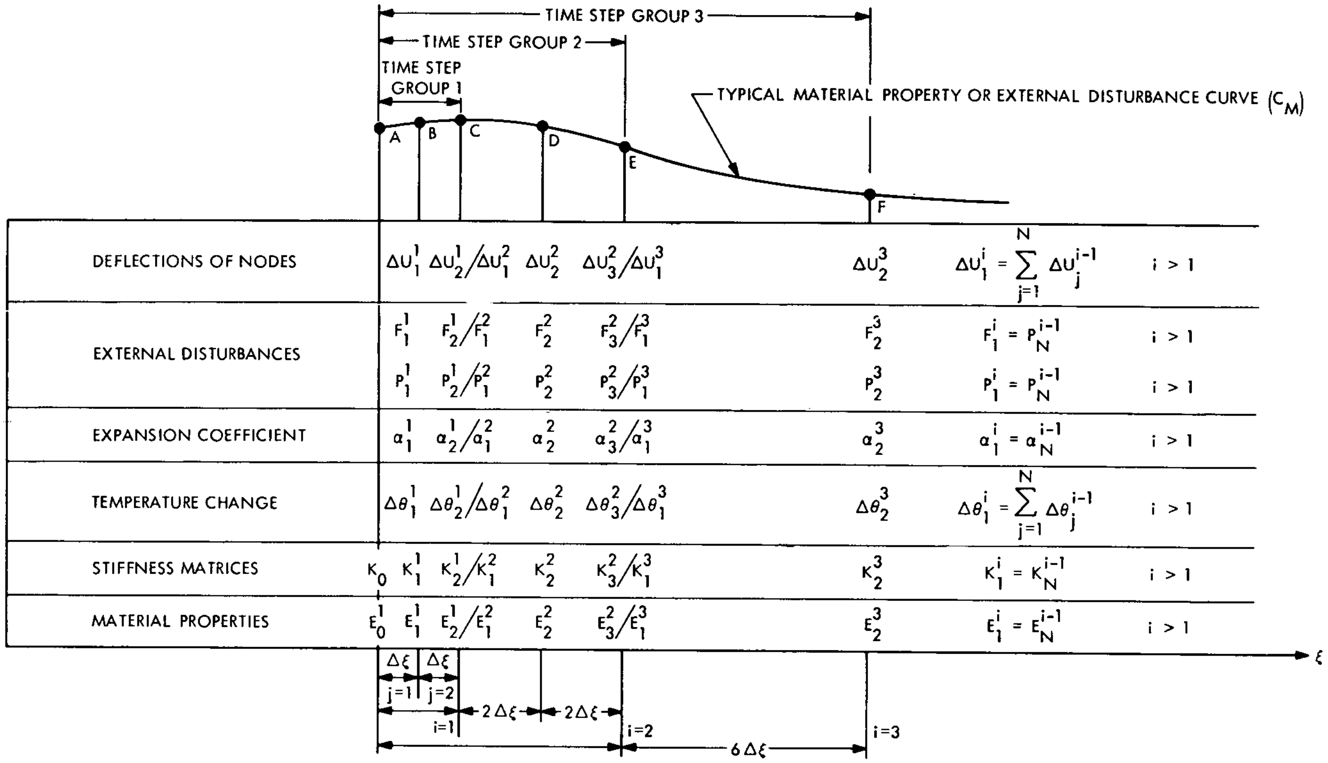


Fig. 2. Typical ξ interval setup

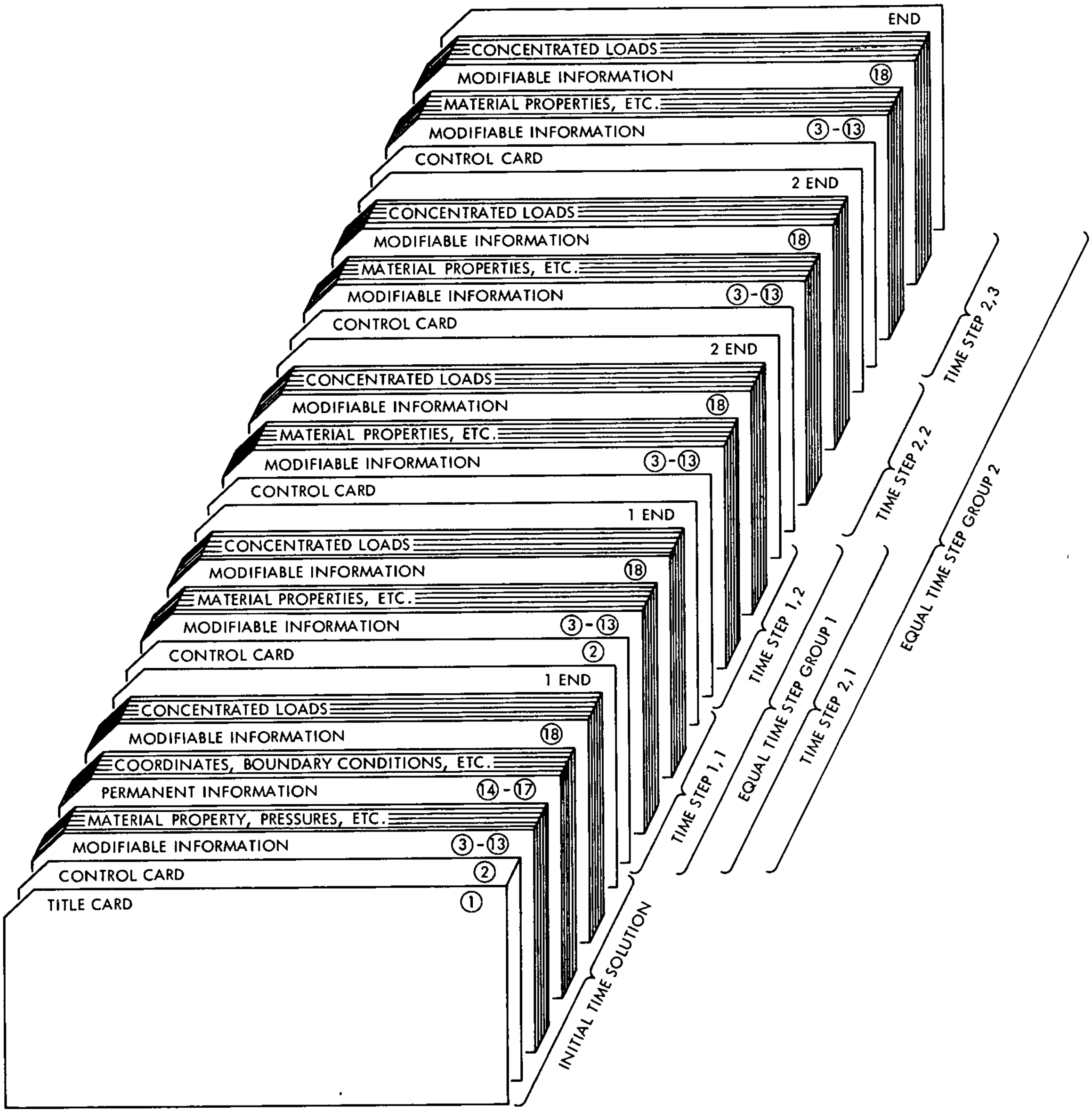


Fig. 3. Physical arrangement of data deck for the VISCEL program

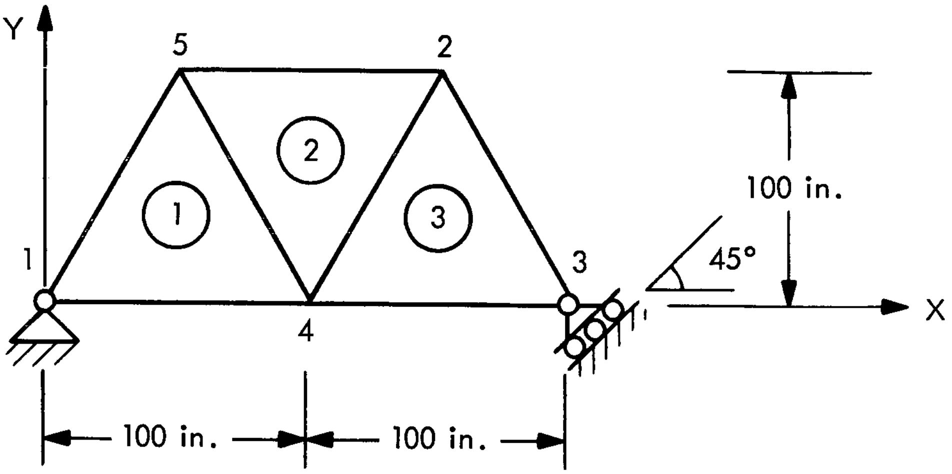
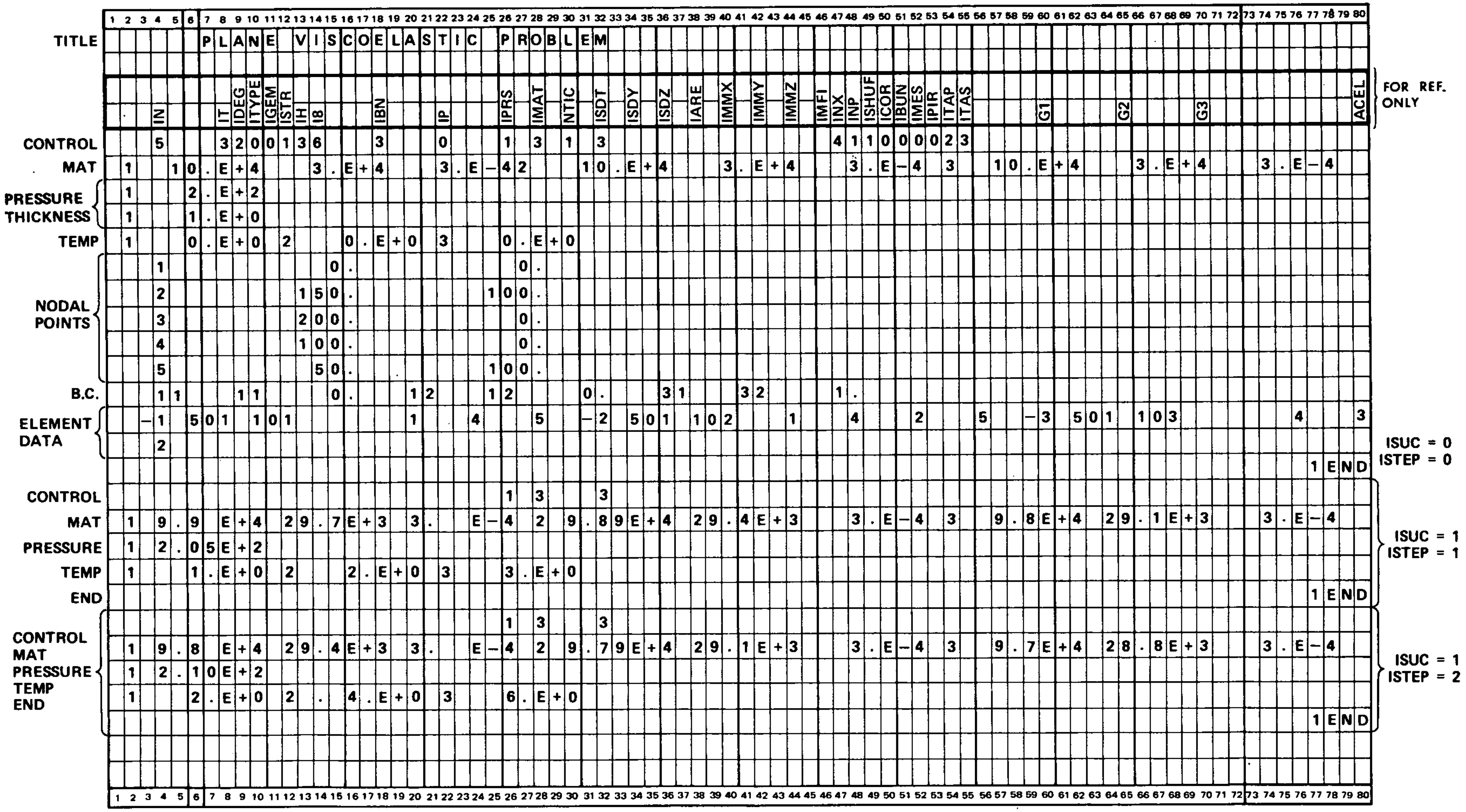


Fig. 4. Plane stress example problem



```
      PLANE VISCOELASTIC PROBLEM
   5   3200136   3   0   1 3 1 3              411000023
 1  10.E+4   3.E+4   3.E-42   10.E+4   3.E+4   3.E-4 3  10.E+4   3.E+4   3.E-4
 1   2.E+2
 1   1.E+0
 1   0.E+0 2   0.E+0 3   0.E+0
   1          0.          0.
   2        150.        100.
   3        200.          0.
   4        100.          0.
   5         50.        100.
   11   11    0.   12   12    0.   31   32    1.
  -1 501 101       1   4   5  -2 501 102   1   4   2   5  -3 501 103       4   3
   2
                                                                            1END
                         1 3   3
 1 9.9 E+4 29.7E+3 3.  E-4 2 9.89E+4 29.4E+3   3.E-4 3  9.8E+4 29.1E+3   3.E-4
 1 2.05E+2
 1   1.E+0 2   2.E+0 3   3.E+0
                                                                            1END
                         1 3   3
 1 9.8 E+4 29.4E+3 3.  E-4 2 9.79E+4 29.1E+3   3.E-4 3  9.7E+4 28.8E+3   3.E-4
 1 2.10E+2
 1   2.E+0 2 . 4.E+0 3   6.E+0
                                                                            1END
```

Fig. 5. VISCEL input data for plane stress example problem

```
CONTROL                           1 3   3
MAT       1 9.7 E+4 29.1E+3 3.  E-4 2 9.69E+4 28.8E+3   3.E-4 3  9.6E+4 28.5E+3   3.E-4  
PRESS.    1 2.15E+2                                                                        ISUC = 1
TEMP      1   3.E+0 2   6.E+0 3   9.E+0                                                    ISTEP = 3
END                                                                                 1END
CONTROL                           1 3   3
MAT       1 9.6 E+4 28.8E+3 3.  E-4 2 9.59E+4 28.5E+3   3.E-4 3  9.5E+4 28.2E+3   3.E-4  
PRESS.    1 2.20E+2                                                                        ISUC = 1
TEMP      1   4.E+0 2   8.E+0 3  12.E+0                                                    ISTEP = 4
END                                                                                 2END
CONT                              1 3   3
MAT       1 9.2 E+4 27.6E+3 3.  E-4 2 9.10E+4 27.5E+3   3.E-4 3  9.0E+4 27.4E+3   3.E-4  
PRESS.    1 2.40E+2                                                                        ISUC = 2
TEMP      1   8.E+0 2  16.E+0 3  24.E+0                                                    ISTEP = 1
END                                                                                 2END
CONTROL                           1 3   3
MAT       1 8.8 E+4 26.4E+3 3.  E-4 2 8.70E+4 26.3E+3   3.E-4 3  8.6E+4 26.2E+3   3.E-4  
PRESS.    1 2.60E+2                                                                        ISUC = 2
TEMP      1  12.E+0 2  24.E+0 3  36.E+0                                                    ISTEP = 2
END                                                                                 0END
```

Fig. 5 (contd)

APPENDIX

VARIOUS REFERENCE TABLES AND FIGURES

Table A-1. Deflection degrees of freedom at a point for different cases of structures

	Column number	1	2	3	4	5	6	7
Case number	Degree-of-freedom direction[a] / Case description	Displacement along X	Displacement along Y	Displacement along Z	Rotation about X	Rotation about Y	Rotation about Z	Number of degrees of freedom
1	Planar truss	▨	▨					2
2	Space truss	▨	▨	▨				3
3	Planar frame	▨	▨				▨	3
4	Space frame	▨	▨	▨	▨	▨	▨	6
5	Gridwork frame			▨	▨	▨		3
6	Plane stress	▨	▨					2
7	Plane strain	▨	▨					2
8	Plate bending			▨	▨	▨		3
9	General solid	▨	▨	▨				3
10	General shell; bend., memb.	▨	▨	▨	▨	▨	▨	6
11	General shell, membrane	▨	▨	▨				3
12	Solid of revolution	▨	▨					2
13	Shell of revolution, membrane	▨	▨					2
14	Shell of rev.; bend., memb.	▨	▨				▨	3

[a] X, Y, Z refer to the axes of the overall coordinate system.

Table A-2. Types of structures that VISCEL can handle

	Case number	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Case number	Description of case	Planar truss	Space truss	Planar frame	Space frame	Gridwork frame	Plane stress[a]	Plane strain[a] ($\varepsilon_z = 0$)	Plate bending	General solid	General shell; bending, membrane	General shell, membrane	Solid of revolution	Shell of revolution, membrane	Shell of revolution; bending, membrane
1	Planar truss	▨					▨	▨							
2	Space truss		▨							▨		▨			
3	Planar frame			▨											
4	Space frame				▨						▨				
5	Gridwork frame					▨			▨						
6	Plane stress[a]	▨					▨								
7	Plane strain[a] ($\varepsilon_z = 0$)	▨						▨							
8	Plate bending					▨			▨						
9	General solid		▨							▨		▨			
10	General shell; bending, membrane				▨						▨				
11	General shell, membrane		▨							▨		▨			
12	Solid of revolution												▨	▨	
13	Shell of revolution, membrane												▨	▨	
14	Shell of revolution; bending, membrane														▨

[a] Cases 6 and 7 may not exist simultaneously.

Table A-3. Element properties

1	2	3	4	5	6	7	8	9	10	11
Element type number	Element geometry	Number of nodes (vertices), IMS	Degrees of freedom per node, IDEG	Number of words for element description, I8	Case No. of structure (Table III-2) for which this element may be used	Nodal line on the first material axis direction	Nodal line or nodal plane on which the pressure may exist	Pressure direction	Orientation of overall coordinate system with respect to structure	Local coordinate system of element
1	Line segment	2	2	5	1	1-2	1-2	$	○	■
1	Line segment	2	3	5	2	1-2	1-2	$	Any	■
2	Line segment	2	3	6	3	1-2	1-2	$	○	□
3	Line segment	2	3	5	5	1-2	1-2	†	○	□
4	Line segment	2	6	8	4	1-2	1-2	$	Any	□
5	Triangle	3	2	6	6, 7	●	1-2	*	○	■
6	Quadrilateral	4	2	7	6, 7	●	1-2	*	○	■
7	Triangle	3	3	6	8	●	1-2-3	**	○	■
8	Quadrilateral	4	3	7	8	●	1-2-3-4	**	○	■
9	Tetrahedron	4	3	6	9	●	1-2-3	**	Any	■
10	Hexahedron	8	3	10	9	●	1-2-3-4	**	Any	■
11	Triangle	3	6	6	10	1-2	1-2-3	**	Any	△
12	Quadrilateral	4	6	7	10	1-2	1-2-3-4	**	Any	△
13	Triangle	3	3	6	11	1-2	1-2-3	**	Any	△
14	Quadrilateral	4	3	7	11	1-2	1-2-3-4	**	Any	△
15	Triangular torus	3	2	6	12	●	1-2	*	◒	■
16	Quadrilateral torus	4	2	7	12	●	1-2	*	◒	■
17	Conical segment	2	2	5	13	1-2	1-2	(*)	◒	▲
18	Conical segment	2	3	5	14	1-2	1-2	(*)	◒	▲

Legend

○	structure is in the overall (X-Y) plane
◒	the mesh is in the overall (X-Y) plane and overall Y axis is the axis of revolution
●	first material axis is the overall X axis
*	normal to nodal line 1-2 and the overall Z axis, and away from element
**	normal to surface shown in column 8 and in direction of local normal
(*)	local z-axis direction
$	Perpendicular to the element in the plane established by the element and the overall X axis. The direction is such that the angle between the perpendicular and the X axis is less than 90 deg
†	in the direction of overall Z axis
■	parallel to overall axes (for element type 1, for stresses, local system as in □)
△	x axis: nodal line 1-2; z axis: normal to middle surface, which sees labels counterclockwise
▲	x axis: nodal line 1-2; y axis parallel and opposite to Z axis
□	x axis: nodal line 1-2; y axis: one of the principal axes of the cross sections

Table A-4. Necessary and optional information for element definition

1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20
Element type number	Node label of first vertex	Node label of second vertex	Node label of third vertex	Node label of fourth vertex	Node label of fifth vertex	Node label of sixth vertex	Node label of seventh vertex	Node label of eighth vertex	Material type number	Thickness type number	Cross-sectional area type number	y moment of inertia type number	z moment of inertia type number	Torsional constant type number	Angle fixing principal directions type number	Pressure type number	Uniform temperature increase type number	Temperature gradient along y type number	Temperature gradient along z type number
1	▒	▒							▒		▒					▨	▨		
2	▒	▒							▒		▒		▒			▨	▨	▨	
3	▒	▒							▒			▒		▒		▨			▨
4	▒	▒							▒		▒	▒	▒	▒	▒	▨	▨	▨	▨
5	▒	▒	▒						▒	▒						▨	▨		▨
6	▒	▒	▒	▒					▒	▒						▨	▨		▨
7	▒	▒	▒						▒	▒						▨			▨
8	▒	▒	▒	▒					▒	▒						▨			▨
9	▒	▒	▒	▒					▒							▨	▨		
10	▒	▒	▒	▒	▒	▒	▒	▒	▒							▨	▨		
11	▒	▒	▒						▒	▒						▨	▨		▨
12	▒	▒	▒	▒					▒	▒						▨	▨		▨
13	▒	▒	▒						▒	▒						▨	▨		
14	▒	▒	▒	▒					▒	▒						▨	▨		
15	▒	▒	▒						▒	▨						▨	▨		
16	▒	▒	▒	▒					▒	▨						▨	▨		
17	▒	▒							▒	▒						▨	▨		
18	▒	▒							▒	▒						▨	▨		▨

▒ Necessary information

▨ Optional information

Table A-5. Types of elements available for different cases of structures

	Column number	1	2	3	4	5	6	7	8
Case number	Case description / Element type	Line segment	Triangle	Quadrilateral	Conical segment	Tetrahedron	Hexahedron	Triangular torus	Quadrilateral torus
1	Planar truss	1							
2	Space truss	1							
3	Planar frame	2							
4	Space frame	4							
5	Gridwork frame	3							
6	Plane stress		5	6					
7	Plane strain		5	6					
8	Plate bending		7	8					
9	General solid					9	10		
10	General shell; bending, membrane		11	12					
11	General shell, membrane		13	14					
12	Solid of revolution							15	16
13	Shell of revolution, membrane				17				
14	Shell of revolution; bending, membrane				18				

Table A-6. Convention for ordering the vertices of elements

Element type number	First vertex	Other vertices
1	Any	The remaining
2	Any	The remaining
3	Any	The remaining
4	Any	The remaining
5	Any	Counterclockwise sequence about overall Z axis
6	Any	Counterclockwise sequence about overall Z axis
7	Any	Counterclockwise sequence about overall Z axis
8	Any	Counterclockwise sequence about overall Z axis
9	Any	Counterclockwise sequence for the first three vertices about the normal of their plane, heading towards the fourth vertex
10	Any	*
11	Any	Clockwise sequence about local normal**
12	Any	Counterclockwise sequence about local normal**
13	Any	Counterclockwise sequence about local normal**
14	Any	Counterclockwise sequence about local normal**
15	Any	Counterclockwise sequence about overall Z axis
16	Any	Counterclockwise sequence about overall Z axis
17	***	The remaining
18	***	The remaining

*Counterclockwise sequence for the first four vertices on the same face about the normal heading towards the other four vertices. The fifth vertex lies diagonally across the first vertex. The last four vertices also establish a counterclockwise sequence about the normal of their face, heading towards the first four vertices.

**Local normals head always to the same side of the space divided by the middle surface.

***The one with smaller meridional arc length (the meridional curve should have a direction).

Table A-7. The functions of the FORTRAN units as used in VISCEL

FORTRAN unit number	Function of the unit
1	System
2	Chain
3	Scratch for topological information generated in Link 3
4	Storage for deflections
5	Input
6	Output
7	Punch
8	Overlays for FORTRAN IV
9	Storage for material elastic constants
10	Storage for material expansion coefficients
11	Storage for temperature changes
12	Storage for elemental stiffness matrices
13	Storage for overall stiffness matrices
14	Scratch for elemental stiffness matrices Scratch for incremental deflections for ISTEP = 1
15	Storage for stiffness matrix decomposed by Choleski scheme

Table A-8. Summary of the problem control card (input item 2) of input data

Name of field	Card columns of field	Format	Range	Description
IN	1–4	I4	2–9999	Total number of mesh points
IT	5–8	I4	1–9999	Total number of elements
IDEG	9	I1	2–6	Number of degrees at a mesh point (see Table A-1, column 7)
ITYPE	10	I1	0–2	Material indicator: 0—isotropic, 1—orthotropic, 2—general (see Fig. A-2)
IGEM	11	I1	0–1	Geometry indicator: IGEM = 0 all Z coordinates are zero[a] (see Table A-3, IGEM = 1 not all Z coordinates are zero[a] column 10)
ISTR	12	I1	0–1	Plane strain case indicator: ISTR = 1 plane strain ISTR = 0 not plane strain
IH	13	I1	2–8	Maximum number of vertices in elements used (see Table A-3, column 3)
I8	14	I1	5–10[b]	Maximum number of words for element description (see Table A-3, column 5)
IBN	15–18	I4	1–9999	Total number of dbc input units (see Section III)
IP	19–22	I4	0–9999	Total number of concentrated load input units (see Section III)
IPRS	23–26	I4	0–99	Total number of different pressures
IMAT	27–28	I2	1–99	Total number of different materials
NTIC	29–30	I2	0–99	Total number of different thicknesses
ISDT	31–32	I2	0–99	Total number of different temperature increases
ISDY	33–34	I2	0–99	Total number of different temperature gradients $(\partial t/\partial y)$[c]
ISDZ	35–36	I2	0–99	Total number of different temperature gradients $(\partial t/\partial z)$[c]
IARE	37–38	I2	0–99	Total number of different cross-sectional areas
IMMX	39–40	I2	0–99	Total number of different torsional constants
IMMY	41–42	I2	0–99	Total number of different moments of inertia (about y axis)[c]
IMMZ	43–44	I2	0–99	Total number of different moments of inertia (about z axis)[c]
IMFI	45–46	I2	0–99	Total number of angles fixing local y and z axes[c]
INX	47	I1	1–4	Number of link after which return-to-beginning-for-next-job is done
INP	48	I1	0–2	Printout indicator: 0—minimum; 1—intermediate; 2—detailed output (see Table A-11)
ISHUF	49	I1	0–3	Relabelling indicator: 0—no relabelling; 1—iterate to relabel without reading cards; 2—read cards and iterate to relabel; 3—relabel as shown on cards (see Ref. 1)
ICOR	50	I1	0–1	Indicator for coordinate generation: 0—read coordinates from cards; 1—generate coordinates via subroutine CORG (user's version) (see Ref. 1)
IBUN	51	I1	0–1	Indicator for displacement boundary conditions: 0—read from cards; 1—generate with user's version of subroutine BUNG (see Ref. 1)
IMES	52	I1	0–1	Indicator for element descriptions: 0—read from cards; 1—generate with user's version of subroutine MESG
IPIR	53	I1	0–2	Local coordinate selection indicator for shells: 0—assume local x as 1-2 line of lowest numbered element; 1—assume as principal; 2—read by subroutine AGEL

Table A-8 (contd)

Name of field	Card columns of field	Range	Format	Description
ITAP	54	I1	0–9	Chain tape number for program (if zero, program assumes 2)
ITAS	55	I1	0–9	Chain tape number for intermediate storage
G1	56–60	F5.4	(−1.)–(+1.)	Cosine of the angle of acceleration vector with X axis[a]
G2	61–65	F5.4	(−1.)–(+1.)	Cosine of the angle of acceleration vector with Y axis[a]
G3	66–70	F5.4	(−1.)–(+1.)	Cosine of the angle of acceleration vector with Z axis[a]
ACEL[d]	71–80	E10.3	Any	Magnitude of acceleration vector times unit mass (unit weight)

[a] X, Y, Z refer to overall coordinate system.

[b] When I8 = 10, zero should be punched in column 14.

[c] x, y, z refer to the local coordinate system of the element.

[d] In element type 3, ACEL means weight per unit length.

Table A-9. Description of element data for different element types

Element type No.	Meanings of variables appearing in the list of Input Item 16 of Table IV-1[a]									
	MM	J1W	J2W	J3W	J4W	J5W	J6W	J7W	J8W	J9W	J10W
1	M	100+IMET	100(JARE)+ITEM	JPRS	N_1	N_2					
2	M	200+IMET	100(JARE)+ITEM	JPRS	100(JMMZ)+JSDY	N_1	N_2				
3	M	300+IMET	100(JPRS)+JSDZ	100(JMMX)+JMMY	N_1	N_2					
4	M	400+IMET	100(JARE)+ITEM	100(JMMX)+JMMY	100(JMMZ)+JSDY	100(JSDZ)+JMFI	JPRS	N_1	N_2		
5	M	500+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3				
6	M	600+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3	N_4			
7	M	700+IMET	100(ITIC)+ITEM	100(JSDZ)+JPRS	N_1	N_2	N_3				
8	M	800+IMET	100(ITIC)+ITEM	100(JSDZ)+JPRS	N_1	N_2	N_3	N_4			
9	M	900+IMET	100(JPRS)+ITEM	N_1	N_2	N_3	N_4				
10	M	1000+IMET	100(JPRS)+ITEM	N_1	N_2	N_3	N_4	N_5	N_6	N_7	N_8
11	M	1100+IMET	100(ITIC)+ITEM	100(JSDZ)+JPRS	N_1	N_2	N_3				
12	M	1200+IMET	100(ITIC)+ITEM	100(JSDZ)+JPRS	N_1	N_2	N_3	N_4			
13	M	1300+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3				
14	M	1400+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3	N_4			
15	M	1500+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3				
16	M	1600+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2	N_3	N_4			
17	M	1700+IMET	100(ITIC)+ITEM	JPRS	N_1	N_2					
18	M	1800+IMET	100(ITIC)+ITEM	100(JSDZ)+JPRS	N_1	N_2					

[a] Definitions:

IMET	material type number	JSDY	temperature gradient along *y* type number	JMFI	angle fixing local *y*, *z* axes type number
ITIC	thickness type number	JSDZ	temperature gradient along *z* type number	N_1	mesh-point label of the first vertex
JPRS	pressure type number	JMMX	torsion constant type number	N_2	mesh-point label of the second vertex, . . . ,
JARE	cross-sectional area type number	JMMY	moment of inertia (about *y*) type number	N_8	mesh-point label of the eighth vertex
ITEM	temperature increase type number	JMMZ	moment of inertia (about *z*) type number	*m*	element label

$M = -\left(m - \left(\frac{m}{1000}\right) * 1000\right)$, which is to be interpreted in FORTRAN integer arithmetic sense

Note: On the cards of Input Item 16 (first option), zero field(s) (of four columns) after the last nonzero field of element description is ignored if provided.

Table A-10. Table for determining the direction of local y axis and the sign of angle ϕ

Parameter	$\lvert \ell_{xX} \rvert > 0.0001$	$\lvert \ell_{xX} \rvert \leq 0.0001$ $\lvert \ell_{xZ} \rvert > 0.0001$	$\lvert \ell_{xX} \rvert \leq 0.0001$ $\lvert \ell_{xZ} \rvert \leq 0.0001$
Positive direction for local y axis	Such that $\ell_{yY} = \cos \phi$	Such that $\ell_{yY} = \cos \phi$	Such that $\ell_{yZ} = \cos \phi$
Sign of ϕ	Negative the sign of $(\ell_{zY}\ell_{xX})$*	Sign of ℓ_{yX}**	Sign of ℓ_{yX}**

*If $(\ell_{zY}\ell_{xX})$ is zero, its sign may be assumed negative.

**If ℓ_{yX} is zero, its sign may be assumed positive.

Table A-11. List of output items

Output item No.	Output item	Minimum INP = 0	Intermediate INP = 1	Detailed INP = 2
	Link 1 (input link)			
1	Table for title and important constants	▨	▨	▨
2	Tables for material, loading, and geometry		▨	▨
3	Table for coordinates of mesh points		▨	▨
4	Table for element properties		▨	▨
5	Message and/or tables and punched cards for relabelling	▨ @	▨ @	▨ @
6	Table for reduced stiffness matrix		▨	▨
7	Message about necessary storage for reduced stiffness matrix	▨	▨	▨
8	Message about total common length	▨	▨	▨
9	Table for diagonal elements of reduced stiffness matrix		▨	▨
10	Tables for force and deflection boundary conditions		▨	▨
11	Table for common, integer (IA block)			▨
12	Table for common, floating (AA block)			▨
13	Message for the execution time of Link 1	▨	▨	▨
	Link 2 (generation link)			
14	Tables for element stiffness matrices	*	*	▨
15	Table for upper half of reduced stiffness matrix	*	*	▨
16	Table for reduced load vector	*	*	▨
17	Message for the execution time of Link 2	▨	▨	▨
	Link 3 (deflection link)			
18	Table for reduced solution vector			▨
19	Table for deflections	▨	▨	▨
20	Table for forces acting on the nodes		▨	▨
21	Message for the execution time of Link 3	▨	▨	▨
	Link 4 (stress link)			
22	Table for stresses at the nodes	▨	▨	▨
23	Details of the best-fit stress computation	†	†	▨
24	Table for stresses of one-dimensional elements	▨	▨	▨
25	Message for the execution time of Link 4	▨	▨	▨

Output produced (blank means no output produced).

@ Output related with relabeling

* May be produced selectively by subroutine CAS2 (see Ref. 1).

† May be produced selectively by subroutine CAS4 (see Ref. 1)

Table A-12. Meanings of the components of stresses at mesh points of two- and three-dimensional continua

Class No.	Element type No.	Structure type	First component	Second component	Third component	Fourth component	Fifth component	Sixth component
1	5, 6	2-D elasticity	σ_1	σ_2	τ_{12}*	0†	0	0
2	7, 8	Plate, bending	0	0	0	M_1	M_2	M_{12}
3	15, 16	Solid of revolution	σ_1	σ_2	σ_3	τ_{12}	0	0
4	9, 10	General solid	σ_1	σ_2	σ_3	τ_{12}	τ_{13}	τ_{23}
5	17	Shell of revolution, membrane	N_1	N_2	0	0	0	0
6	18	Shell of revolution, membrane and bending	N_1	N_2	0	M_1	M_2	0
7	13, 14	Shell, membrane	N_1	N_2	N_{12}	0	0	0
8	11, 12	Shell, membrane and bending	N_1	N_2	N_{12}	M_1	M_2	M_{12}

[a]Nomenclature:

σ_1, σ_2, σ_3 normal stresses

τ_{12}, τ_{13}, τ_{23} shear stresses

N_1, N_2 membrane normal forces

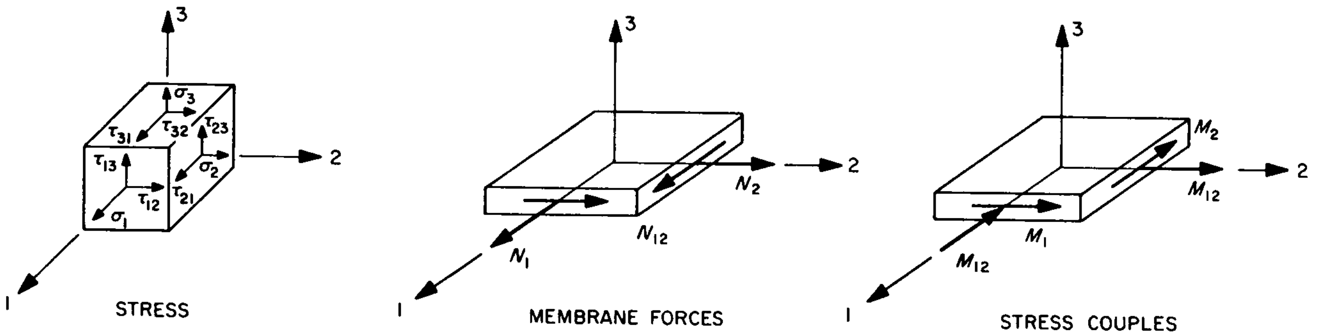
N_{12} membrane shear

M_1, M_2 bending stress couples

M_{12} twist stress couple

*σ_3 for plane-strain case

†τ_{12} for plane-strain case



The axis labels 1, 2, and 3 stand for KSI, ETA and ZTA (direction cosines for local axes), respectively

Table A-13. List of error messages

No.	Error message
1.	INPUT ERROR
2.	THE FOLLOWING DISPLACEMENT BOUNDARY CONDITION(S) CAUSE(S) MORE THAN ONE MULTIPLE CONNECTION FOR THE UNKNOWNS. THEY ARE IGNORED
3.	*i*: IN ELEMENT . . . , ERROR IN MESH TOPOLOGY INFORMATION. NO CORRECTION IS MADE. *ii*: IN ELEMENT . . . , . . . PROPERTY TYPE NUMBER(S) IS OUTSIDE THE RANGE. THE TYPE NUMBER(S) IS ASSUMED LARGEST POSSIBLE
4.	ELEMENT . . . IS UNACCEPTABLE. DISREGARDED
5.	WARNING. LESS THAN 12750 DECIMAL LOCATIONS ARE AVAILABLE FOR THE NEXT LINK PROGRAMS. THOUGH IT MAY BE SUICIDAL, EXECUTION CONTINUES
6.	THE POINT . . . DOES NOT APPEAR IN THE MESH TOPOLOGY
7.	DUMMY AREA OVERLAPS COMMON AREA BY . . . DECIMAL LOCATIONS. RECOMPILE BY CHANGING THE EQUIVALENCES OF DUMMY AND BB IN LINKS 1 AND 3, RESPECTIVELY
8.	ELEMENT . . . , . . . IS UNACCEPTABLE. DISREGARDED . . .
9.	THE VOLUME OF ELEMENT . . . , . . . IS TOO SMALL . . . DISREGARDED
10.	STIFFNESS MATRIX IS NOT POSITIVE DEFINITE . . .
11.	NO SCRATCH TAPE IS GIVEN OR ERROR IN SCRATCH TAPE
12.	MORE THAN 12 NON-ONE-DIMENSIONAL ELEMENTS AT NODE . . .
13.	NODAL STRESS COMPUTATION IS DELETED DUE TO PRECEDING
14.	NO SCRATCH TAPE. STRESS LINK IS NOT EXECUTED
15.	ERROR IN READING ELEMENT SETS FROM TAPE ITAS. STRESS LINK EXECUTION IS DELETED . . . , . . .
16.	NOT ENOUGH INDEPENDENT INFORMATION AVAILABLE
17.	ERROR IN MESH TOPOLOGY. NODE ASSUMED INTERNAL
18.	MORE THAN 4 MATERIALS, FIRST 4 CONSIDERED
19.	MORE THAN 4 CLASSES, FIRST 4 CONSIDERED
20.	MORE THAN 19 ELEMENTS, FIRST 19 CONSIDERED
21.	NOT ENOUGH INFORMATION FOR BEST-FIT QUADRATIC BEST-FIT PLANE IS USED
22.	NOT ENOUGH INFORMATION FOR MIDDLE SURFACE NORMAL. APPROXIMATE XII AND ZTA VALUES ARE USED
23.	SCRATCH AREA FF OVERLAPS WITH RESIDUAL AREA. PUSH FF FURTHER DOWN BY RECOMPILING LINK 4.

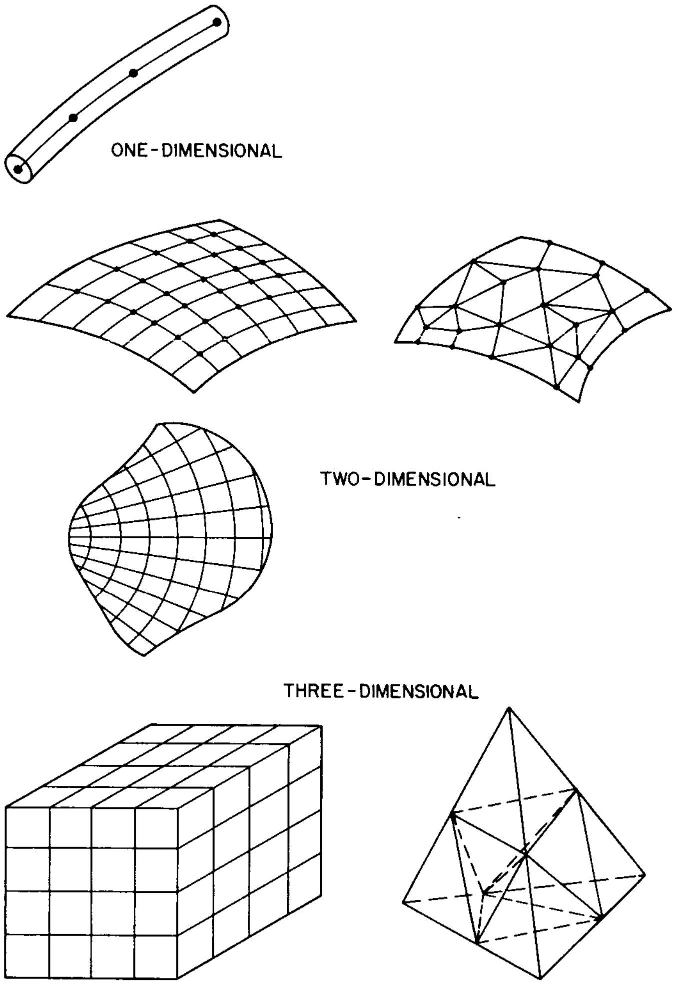
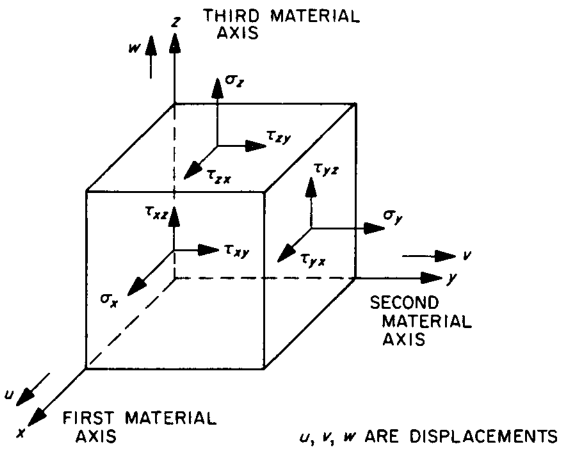


Fig. A-1. One-, two-, and three-dimensional finite element meshes



$$\varepsilon_x = \frac{\partial u}{\partial x},\ \varepsilon_y = \frac{\partial v}{\partial y},\ \varepsilon_z = \frac{\partial w}{\partial z},\ \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x},$$

$$\gamma_{xz} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z},\ \gamma_{yz} = \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}$$

(a)

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} & D_{15} & D_{16} \\ & D_{22} & D_{23} & D_{24} & D_{25} & D_{26} \\ & & D_{33} & D_{34} & D_{35} & D_{36} \\ & & & D_{44} & D_{45} & D_{46} \\ & \text{SYM} & & & D_{55} & D_{56} \\ & & & & & D_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$$

$\{\sigma\} = [D]\{\varepsilon\},\ [D]$: MATERIAL MATRIX

$\{\alpha\} = \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_z \end{Bmatrix},\ \{\alpha\}$: VECTOR OF THERMAL EXPANSION COEFFICIENTS

(b)

ISOTROPIC CASE

$$[D_0] = \begin{bmatrix} aE & bE & bE & 0 & 0 & 0 \\ & aE & bE & 0 & 0 & 0 \\ & & aE & 0 & 0 & 0 \\ & & & G & 0 & 0 \\ & \text{SYM} & & & G & 0 \\ & & & & & G \end{bmatrix}$$

$$a = \frac{G\,(4G-E)}{E\,(3G-E)},\ b = \frac{G\,(2G)}{E\,(3G-E)}$$

$[\alpha] = [\alpha, \alpha, \alpha]$

INPUT: E, G, α

(c)

ORTHOTROPIC CASE

$$[D_1] = \begin{bmatrix} D'_{11} & D'_{12} & * & D'_{14} & 0 & 0 \\ & D'_{22} & * & D'_{24} & 0 & 0 \\ & & \dagger & 0 & 0 & 0 \\ & & & D'_{44} & 0 & 0 \\ & \text{SYM} & & & D'_{55} & D'_{56} \\ & & & & & D'_{66} \end{bmatrix}$$

* IN PLANE-STRAIN CASE D_{12}, OTHERWISE 0

† IN PLANE-STRAIN CASE NOT NECESSARY, OTHERWISE 0

$[\alpha] = [\alpha'_1, \alpha'_2, 0]$

INPUT: $D'_{11}, D'_{12}, D'_{14}, D'_{22}, D'_{24}, D'_{44}, D'_{55}, D'_{56}, D'_{66}$
α'_1, α'_2

(d)

GENERAL CASE

$$[D_2] = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} & D_{15} & D_{16} \\ & D_{22} & D_{23} & D_{24} & D_{25} & D_{26} \\ & & D_{33} & D_{34} & D_{35} & D_{36} \\ & & & D_{44} & D_{45} & D_{46} \\ & \text{SYM} & & & D_{55} & D_{56} \\ & & & & & D_{66} \end{bmatrix}$$

$[\alpha] = [\alpha_1, \alpha_2, \alpha_3]$

INPUT: $D_{11}, D_{12}, D_{13}, D_{14}, D_{15}, D_{16}, D_{22},$
$D_{23}, D_{24}, D_{25}, D_{26}, D_{33}, D_{34}, D_{35},$
$D_{36}, D_{44}, D_{45}, D_{46}, D_{55}, D_{56}, D_{66}$
$\alpha_1, \alpha_2, \alpha_3$

(e)

Fig. A-2. Description of the material

NASA — JPL — Coml., L.A., Calif.