IMPLEMENTATION ON A NONLINEAR CONCRETE CRACKING ALGORITHM IN NASTRAN

David N. Herting, David L. Herendeen and Richard L. Hoesly
Universal Analytics, Inc.

H. Chang
Ebasco Services Incorporated

SUMMARY

A computer code for the analysis of reinforced concrete structures has been developed using NASTRAN as a basis. Nonlinear iteration procedures were developed for obtaining solutions with a wide variety of loading sequences. A direct access file system was used to save results at each load step to restart within the solution module for further analysis. A multi-nested looping capability was implemented to control the iterations and change the loads. The basis for the analysis is a set of multi-layer plate elements which allow local definition of materials and cracking properties.

INTRODUCTION

Faced with an ever-growing need to ensure environmental safety, the nuclear industry must develop analytical tools to determine the physical integrity of reactor structures. These structures, typically layers of concrete interspersed with reinforcing materials, must withstand severe emergency loading conditions. Resulting high stresses due to these loads are expected to cause the formation and propagation of cracks within the concrete. Once this process begins, the structural characteristics change. Thus, a nonlinear solution algorithm is required with the capability of subjecting the structure to an arbitrary sequence of loadings -- thermal, pressure, gravitational and equivalent dynamics. Also, the algorithm designed must be efficient for the solution of large-order and complex structural models.

Although the present NASTRAN (NASA Structural Analysis) computer program system contains a limited elasto-plastic analysis capability in Rigid Format 6 (Piecewise Linear Analysis), it was unacceptable for simulating the three-dimensional concrete cracking phenomena. The present NASTRAN stress-strain rules for nonlinear materials do not allow for a decrease in stress with increasing strain. This effect is certainly evidenced by the behavior of concrete where the internal loads disappear due to cracking. Furthermore, the present analysis is limited to a stepwise application of a single load vector and requires large amounts of computer time.

This paper presents the implementation in Level 15.5 NASTRAN of a nonlinear concrete analysis capability. The effort, performed by Universal Analytics, Inc. (UAI) and Ebasco, involved the addition of nonlinear elements, new functional modules, a DMAP (Direct Matrix Abstraction Program) alter package, and modifications to the NASTRAN executive system. This project illustrates the practicality of modifying NASTRAN for a specialized problem in contrast to the development of general capabilities as implemented in the standard NASA versions.

The basic computational methods were chosen to provide efficient solutions to a wide variety of concrete cracking problems. Multi-layer quadrilateral and triangular plate elements with independent layer-by-layer material definition were implemented to calculate both linear and nonlinear stiffness matrix and load effects.

A nonlinear iteration system was implemented to obtain solutions for large-order problems from efficient vector processing techniques. The system, controlled by DMAP, is controlled by a new module which performs nonlinear loading functions, convergence testing, and automated property updates (for design optimization).

For user convenience in the practical solution of complex structural problems, several modifications were implemented in the NASTRAN Executive system. A restart capability for re-analysis of a problem starting at any previous solution was performed with the addition of a random access storage file independent of the NASTRAN file system. New DMAP language modifications, currently in Level 16 NASTRAN, were installed to provide data block control for multinested DMAP looping.

In the following discussion, the physical characteristics of the expected structures are described, followed by separate descriptions of the major items.

PHYSICAL PROBLEM

Many types of composite steel/concrete construction methods are used in the fabrication of high-strength structures. The hypothetical example shown in figure 1 illustrates the nature of a containment vessel used in nuclear power plant construction. Each layer represents a different property which may contain one or more types of materials. For example, a layer of closely spaced thin rods is imbedded in a concrete matrix and the stress-strain properties represent the effects of both materials as a composite. The combination of bending and membrane forces in the structure contribute nonuniform stresses through the depth of the plate and therefore require a layer-by-layer analysis of the nonlinear effects.

The nonlinear effects of concrete cracking are significantly different physically from the effects of plastic strains in metal structures. Classical elasto-plastic analysis methods use one or more plastic "flow rules" which assume a continuous transition between elastic and plastic behavior with stresses always increasing with increasing strains (refs. 1 and 2). However, the concrete stresses can be assumed linear until cracking occurs, at which point the stresses normal to the crack must be completely removed, leading to problems with discontinuities in the general solution.

The theoretical limits for the principal strains in concrete are shown in figure 2, along with the actual approximations used. Also shown in the stress-strain diagram for a principal strain. The stresses, and therefore the resulting internal forces, will be discontinuous and may lead to difficulties in the solution algorithm. This problem is discussed in a subsequent section.

REINFORCED CONCRETE ELEMENTS

New triangular and quadrilateral plate elements were developed to represent the layered properties of the reinforced concrete. The generalized displacements and element geometry are identical to the existing NASTRAN TRIA1 and QUAD1 plate elements. However, the nonlinear, multi-layered properties required a reformulation of the basic equations to account for the resulting coupling between the in-plane membrane and bending forces. A basic approach similar to the method used in reference 3 was adapted for use with NASTRAN-type element formulations.

Each layer of the new triangular element (TRCA) illustrated in figure 3 corresponds to a unique concrete or steel property which may be defined independently. The grid point locations may be off-set from the element reference coordinate system and the mean bending plane of the element does not require user calculation.

In existing NASTRAN plate elements the membrane and bending properties are calculated independently. However, the coupling terms occurring in the layered elements require their simultaneous calculations. For these new elements, the existing NASTRAN bending routines have been expanded to include in-plane displacements and strains for each component basic triangle. Since the triangle and quadrilateral elements in NASTRAN are comprised of component triangles, the combination process also was expanded to include the coupled in-plane terms.

During solution, the basic element calculations are used for several purposes. These include:

1. Generation of a new stiffness matrix K to represent the current cracked condition.

2. Generation of loads due to thermal expansion with forces across cracked layers removed.

3. Generation of corrective forces P for each element representing the change in forces due to new cracks at a particular deformed shape.

4. Calculation of stresses in the cracked layers for output processing.

5. If requested, the thicknesses of cracked layers may be automatically modified to eliminate the cracking strains. This option is used for design optimization.

The finite elements which may include the standard NASTRAN elements are assembled into stiffness and mass matrices. Multipoint and single-point constraints may be used to define boundaries. The static solution to applied loads, enforced displacements and temperatures is described in the next section.

SOLUTION METHOD

With finite elements, the general form of the static nonlinear equation of state is

$$\{\Phi(u,T)\} = \{P_e\} \tag{1}$$

where $\{\Phi\}$ is the generalized load vector resulting from element stresses

u are the set of node point displacements

T is the temperature field

$\{P_e\}$ is the vector of externally applied loads at the node points

The load function vector, Φ, may be obtained from the total energy of the system, U, and Lagrange's equation, or:

$$\Phi_i = \frac{\partial U}{\partial u_i} \tag{2}$$

For the concrete cracking problem, the system may be reduced to a bounded linear problem. At any cracking condition, i, the generalized loads are:

$$\{\Phi\} = [K_i]\{u_i\} - \{Q_i(T)\} \tag{3}$$

where $[K_i]$ is the stiffness matrix and $\{Q_i\}$ is the thermal load vector.

To be valid, the displacements $\{u_i\}$ in the above equation must be compatible with the cracking state used in $[K_i]$. To find the displacements, the following procedure is used. Equation (1) may be so structured as to produce the iteration equation below:

$$[K_a]\{u_{i+1}\} = \{P_e\} - \{\Phi(u_i)\} + [K_a]\{u_i\} \tag{4}$$

where the index i indicates each step of the iteration and each vector $\{u_{i+1}\}$ is a new estimate of the displacement.

Two choices for the matrix $[K_a]$ are available in the iteration process. In the Newton-Raphson method (ref. 4) the matrix $[K_a]$ is updated at each step using the new displacement vector, $\{u_{i+1}\}$. In the so-called perturbation method, a constant stiffness matrix $[K_a]$ may be used, thereby eliminating the need to decompose a new matrix at each time step. The perturbation method is used initially. If the measured convergence rate is too slow, the matrix is updated to speed up the convergence.

If $[K_a]$ is the matrix corresponding to the initial cracking condition $\{u_i\}$, the vector $\{\Phi\}$ from equation (3) may be substituted into equation (4) to produce the equation:

$$[K_a]\{u_{i+1}\} = \{P_e\} + \{Q_i(T)\} + [K_a - K_i]\{u_i\} \tag{5}$$

Note that the stiffness matrix $[K_a]$ is constant, and no matrix decompositions are required except for the first step.

For calculation purposes, equation (5) is rearranged to produce the actual iteration equation:

$$[K_a]\{u_{i+1}\} = \{P_e\} + \{Q_a\} + \{\delta P_i\} \tag{6}$$

where

$$\{\delta P_i\} = [K_a - K_i]\{u_i\} + \{Q_i - Q_a\} \tag{7}$$

Note that the variable quantity on the right-hand side is the "corrective load" $\{\delta P_i\}$. This is a function only of the elements which have changed in stiffness between states a and i. An exact solution is obtained when a displacement vector is found which satisfies equation (1), or if $\{u_{i+1}\} = \{u_i\}$.

In the above method, the initial stiffness matrix $[K_a]$ may be obtained from any previous cracking condition or may be updated in the iteration procedure. The starting vector ($i = a$) must be chosen such that no additional cracks will be formed with respect to state a. For this reason, the initial starting vector (u_i at $i = a$) is always null.

The iteration procedure is illustrated in figure 4 for a single degree of freedom problem with several layers of steel and concrete. The sawtooth curve represents the force-displacement function, Φ. Each peak represents the limit of each layer and the dashed line represents the applied load. Note that several solutions exist (where the dotted line intersects the solid lines). If the structure was initially uncracked, the iterations would follow the dotted lines. However, once a layer is cracked, the unloading curve follows a different path, i.e., corresponding to the stiffness matrix of the cracked structure. New applied loads will also use this path.

Both convergence and divergence of the system may be tested after each iteration. The loading error vector $\{\delta_i\}$ is defined as:

$$\{\delta_i\} = \{\Phi_i\} - \{P_e\} \tag{8}$$

and from equations (3) and (7):

$$\{\delta_i\} = [K_i]\{u_i\} - \{Q_i\} - \{P_e\}$$

$$= -\{\delta P_i\} + [K_a]\{u_i\} - \{Q_a\} - \{P_e\} \tag{9}$$

Substituting equation (6) for the second term on the right side, we obtain:

$$\{\delta_i\} = \{\delta P_{i-1}\} - \{\delta P_i\} \tag{10}$$

The energy error of the system is obtained from the vector products of displacement and loads. As shown in figure 4 (i = 3), the load error may diverge for one step, yet the process may still converge. In practice, however, for large-order problems, the curves Φ become more continuous and the temporary type of divergence is rarely encountered.

Many of the above matrix operations could be processed by existing NASTRAN DMAP as with the similar Level 16 differential stiffness procedure. However, because nonlinear element routines may be designed to directly calculate the corrective loads defined in equation (7), and because most of the vector operations may be performed more efficiently in core, the basic iterations are performed in a new NASTRAN module. Described below is the basic organization of the NASTRAN implementation.

NASTRAN IMPLEMENTATION

The implementation of the concrete analysis (CA) algorithm in NASTRAN required only isolated interfaces to the program due to NASTRAN's modular design. The interfaces of the new elements were straightforward due to the organization of the element table processing. Most of the new solution iteration procedure code was isolated to six new modules which used many of the existing NASTRAN matrix subroutines.

The cracking analysis solution algorithm was implemented using the NASTRAN DMAP (Direct Matrix Abstraction) control language. The Level 16 DMAP compiler (ref. 5), available in Level 16 NASTRAN, was used to take advantage of its improved capabilities and constructs not available in the previous compiler. The most important of these features are the utility module SWITCH and the freedom to use Data Blocks before they are defined. SWITCH allows Data Block names to be interchanged so that two successive iterates may be SWITCHed and a branch made back to the iteration module.

The initial framework for the cracking analysis DMAP used the Piecewise Linear DMAP of Rigid Format 6. DMAP ALTERs were inserted as required to implement the new technique. The overall logical flow of the DMAP, paying particular attention to the relationship of the new modules and loop controls, is shown in figure 5. The basic computational procedures for the cracking analysis are carried out by the six new modules that were implemented. A brief description of each appears below.

1. Functional Module CASMG (Strain Matrix Generator)

   Generates the cracking analysis strain matrix which will transform grid point idsplacements into generalized element strain of the element. This strain matrix is stored to be used within the iteration loop to compute:

   a. Thermal loads
   b. Element stress
   c. Corrective load vectors

2. Functional Module CAUTIL (Cracking Analysis Utilities)

   Performs several utility functions for the cracking analysis. These include:

   a. Extracting data from the CAØF (defined in a later section) for previous material states
   b. Appending interpolated element temperatures to the EST
   c. Reorganizing property change specification Bulk Data
   d. Assembling partitioning vectors for CAITER (described below)

3. Functional Module CASMA (Element Stiffness Assembler)

   Calculates the variable stiffness matrix for the cracking analysis. This matrix is assembled for each element on the initial pass, and only for updated elements on subsequent passes.

4. Functional Module CAECPT (Cracking Analysis ECPT)

   Generates the updated ECPTCA data block with all element property updates for the current iteration. The updated ECPTCA is also saved on the CAØF. In addition, if DIAG 25 is set, a summary of all elements that cracked in the last stiffness iteration will be printed.

5. Functional Module CAITER (Crack Analysis Iteration)

   Iterates to an equilibrium state solution for the nonlinear cracking analysis.

6. Functional Module CAØUTP (Cracking Analysis Output Processor)

   Formats and prints the output data for a cracking analysis, including stresses, forces and property updates.

Module CAITER is the primary iteration module. It controls the corrective load iteration, property updates, convergence testing, and setting of parameters to control DMAP execution. The logical flow is shown in figure 5.

In order to implement the desired solution paths of the cracking analysis, a random access 'Cracking Analysis Operating File' (CAØF) was integrated into the solution methodology. The standard NASTRAN data base, GINØ, could not adequately handle the required procedures. The CAØF is a stylized version of the Automated Multi-Stage Substructuring Operation File (SØF) (ref. 6) found in standard Level 16 NASTRAN. This file allows the user maximum versatility during solution, the capability to use previous subcases as initial conditions for subsequent cases, and the cost effectiveness of internal restart.

Data for each cracking element and each subcase is stored on the CAØF until deleted by the user. For elements, the stiffness and strain matrices are stored on the CAØF to allow rapid assembly of system matrices similar to Level 16 NASTRAN. For each subcase, the current element summary (EST) and connection (ECPT) tables are stored. Also stored are the most recent load and displacement vectors which may represent a final or some intermediate stage in the iteration cycle. Current iteration parameters are also saved at each step. In the event that convergence has not been obtained, or a system failure has occurred, these parameters may be used to reinitiate the solution. This particular powerful tool allows an 'internal' restart to be performed either at some stage in the solution DMAP or within the CAITER module itself, outside the standard NASTRAN restart framework.

CONCLUSIONS

The nonlinear concrete cracking analysis described above has been implemented in a Level 15.5 version of NASTRAN. Although extensive testing has been performed using moderate size problems, the program is still being tested with large-order problems to provide experimental correlations. With the basic cracking analysis capabilities and the additional user conveniences for multiple cases, the program is expected to become a powerful tool for the analysis of reinforced concrete structures.

Many of the program techniques used in the concrete analysis modifications may be adapted to other types of NASTRAN analyses. A capability to perform automatic internal restarts inside a module, which could be implemented in other NASTRAN modules, has been proven. New methods of processing nonlinear elements have been implemented to provide additional efficiencies. New, highly efficient iteration techniques have been installed which combine core-held vector iterations in a module with out-of-core matrix iterations using DMAP. These operations could be adapted to perform more general types of nonlinear structure analysis such as elasto-plastic analysis combined with crack elements and nonlinear geometry behavior.

REFERENCES

1. Swedlow, J. L.: The Thickness Effect and Plastic Flow in Cracked Plates. Aerospace Research Laboratories, Wright Patterson Air Force Base, Ohio, ARL 65-216, October 1965.

2. Hutchinson, J. W.: Finite Strain Analysis of Elastic-Plastic Solids and Structures, Numerical Solution of Nonlinear Structure Problems. ASME-AMD-Vol. 6, R. F. Hartung, Ed., November 1973.

3. Hand, F. R.; Pecknold, D. A.; and Schnobrick, W. C.: Nonlinear Analysis of RC Plates and Shells. ASCE Journal of the Structure Division, July 1973.

4. Stricklin, J. A., et al.: Survey of Static Geometric and Material Nonlinear Analysis by the Finite Element Method. Proceedings of 2nd U.S.-Japan Seminar on Matrix Methods in Structure Mechanics, University of Alabama Press, 1972.

5. Herendeen, D. L.: An Improved DMAP Capability. NASTRAN: Users' Experiences, NASA TM X3278, 1975.

6. Field, E. I., et al.: The Automated Multi-Level Substructuring System for NASTRAN. NASTRAN: Users' Experiences, NASA TM X3278, 1975.

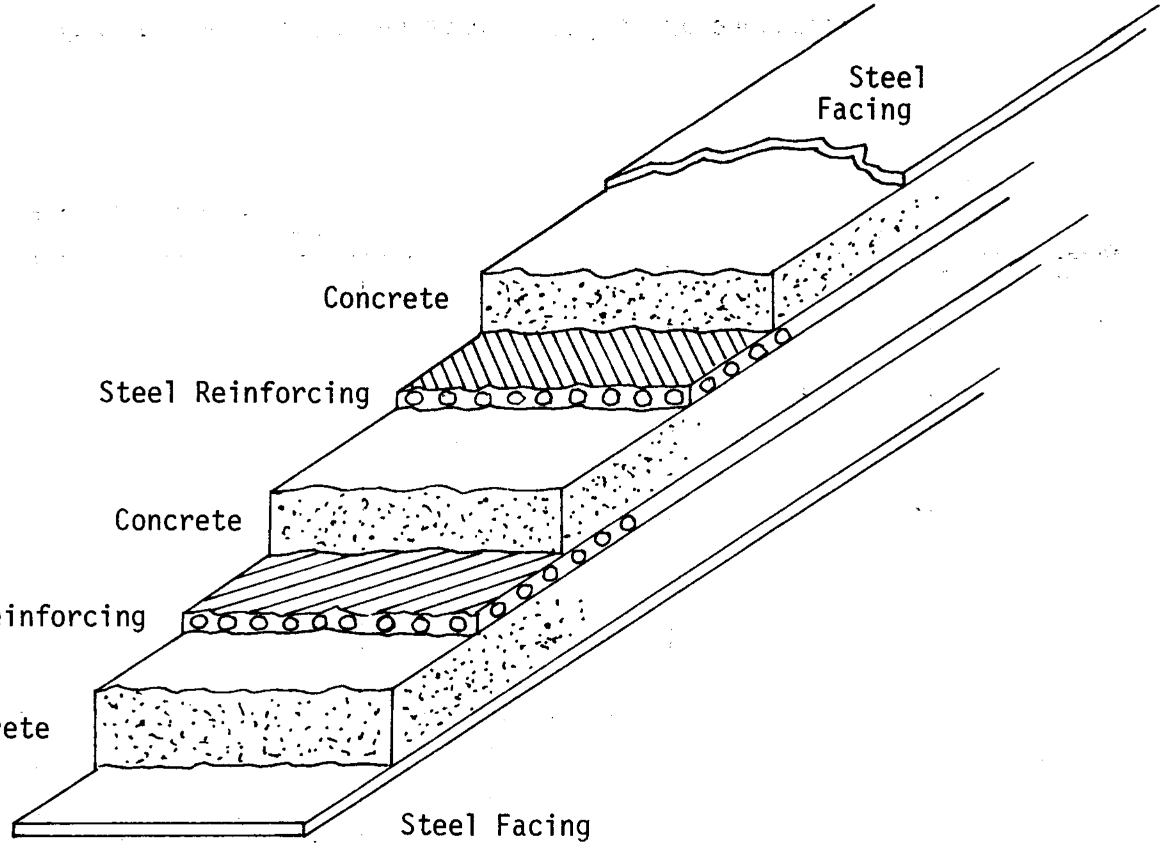

Figure 1.- Sample multi-layer reinforced concrete.

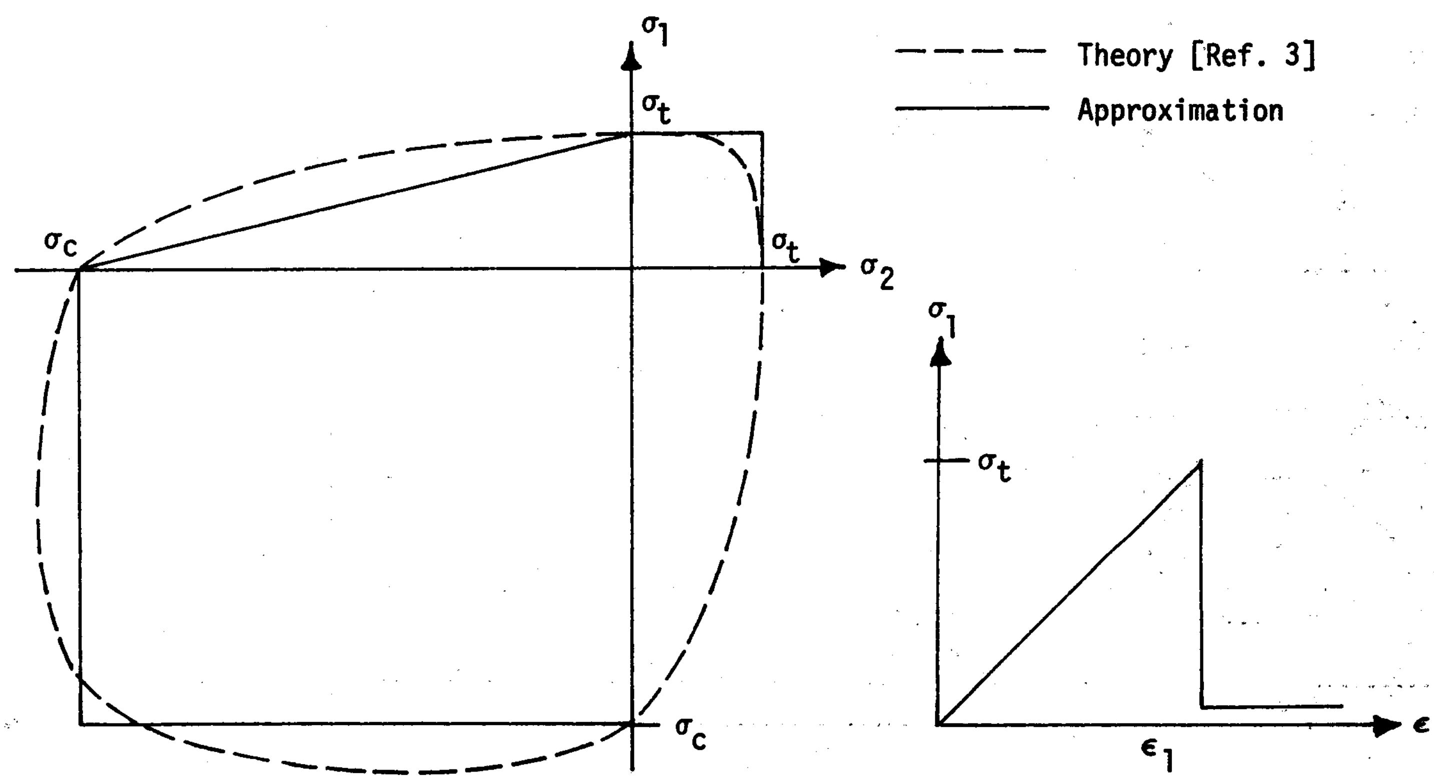

Figure 2.- Stress and strain limits for concrete.

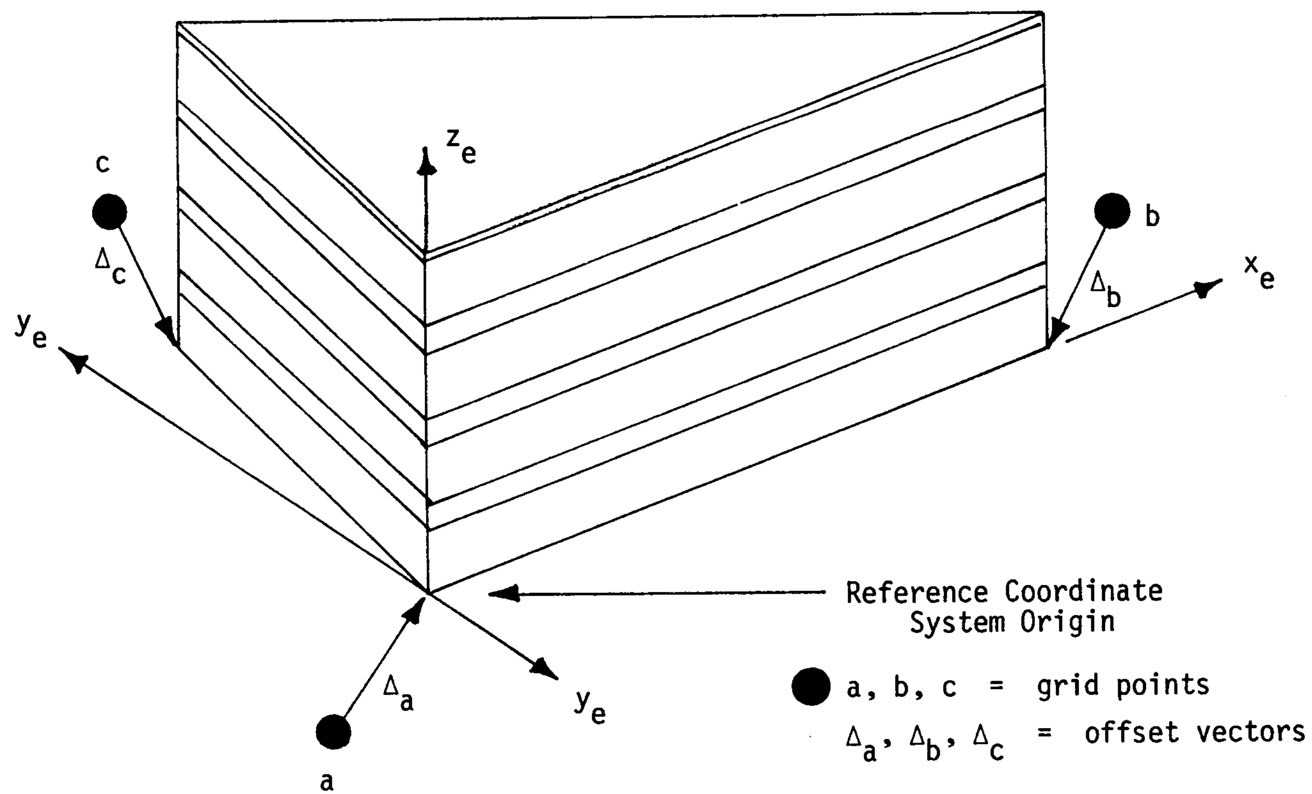

Figure 3.- Triangular crack analysis plate element.

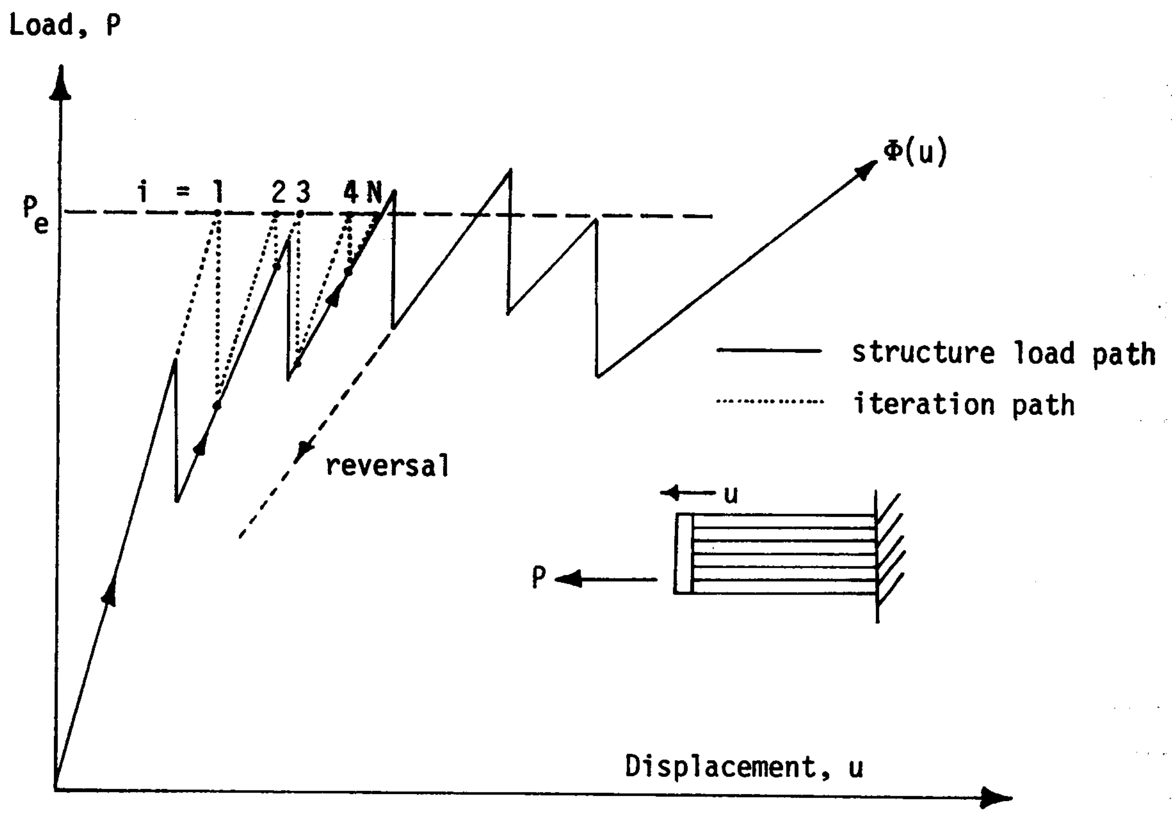

Figure 4.- Load functions for single multi-layer element.

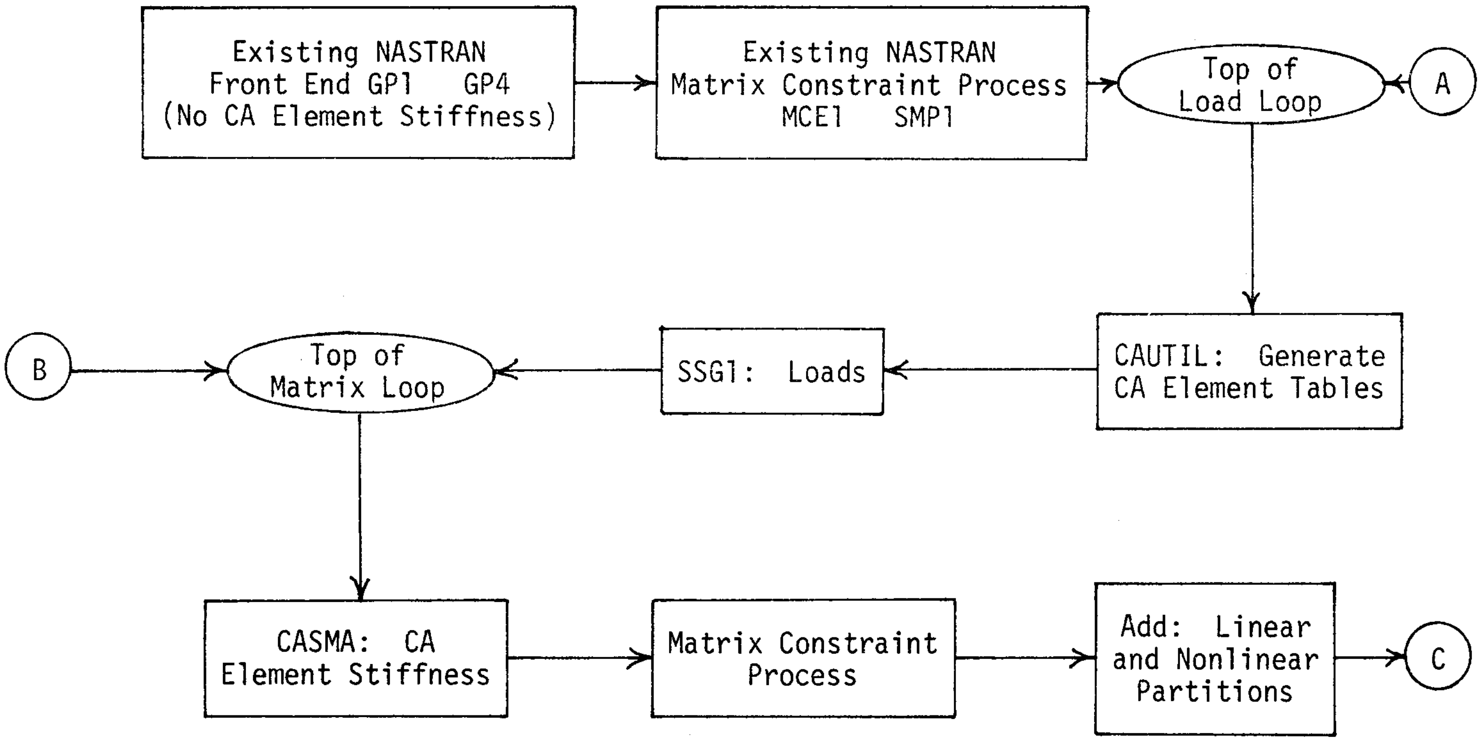

Figure 5.- Logical flow for concrete analysis in NASTRAN.

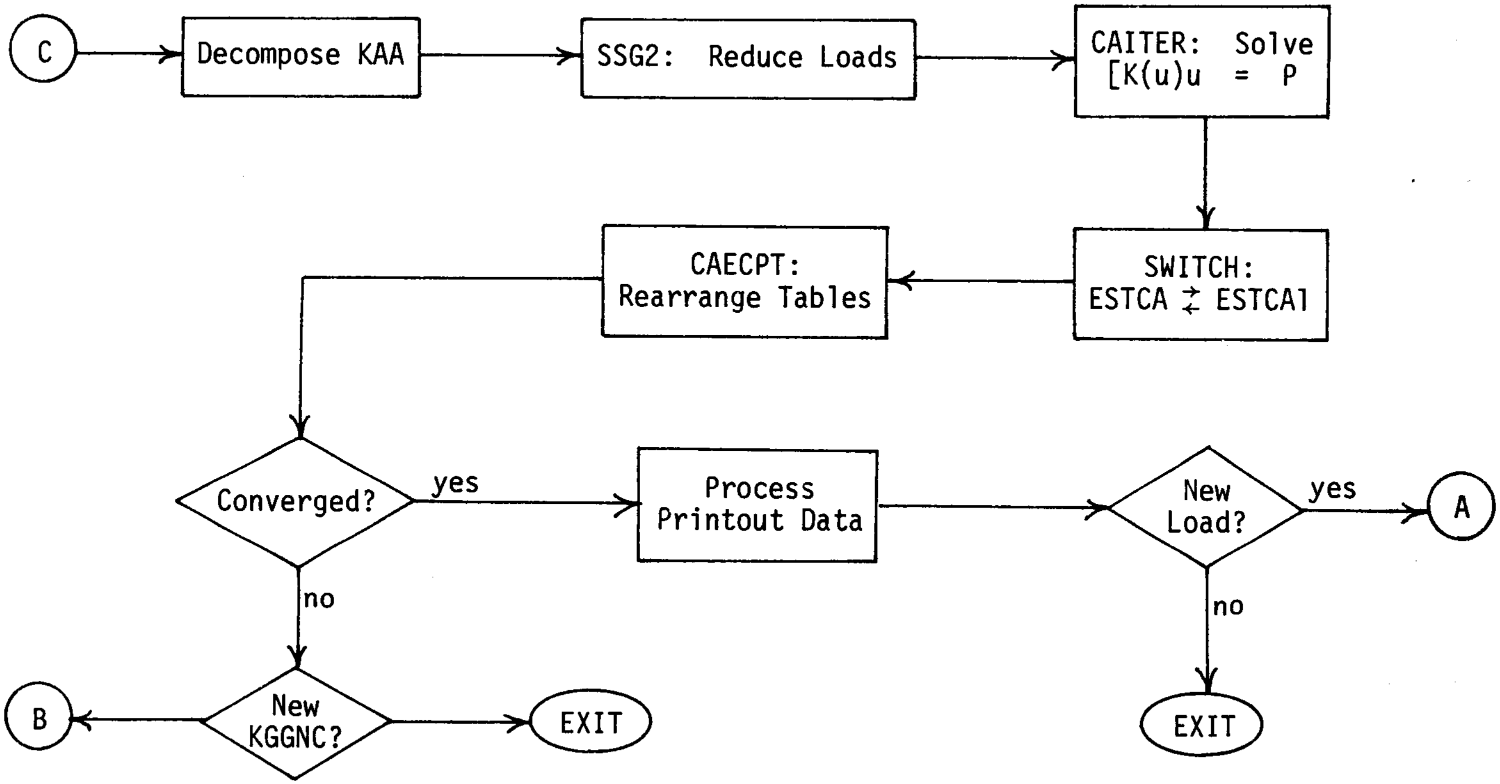

Figure 5.- Logical flow for concrete analysis in NASTRAN. (Cont'd)

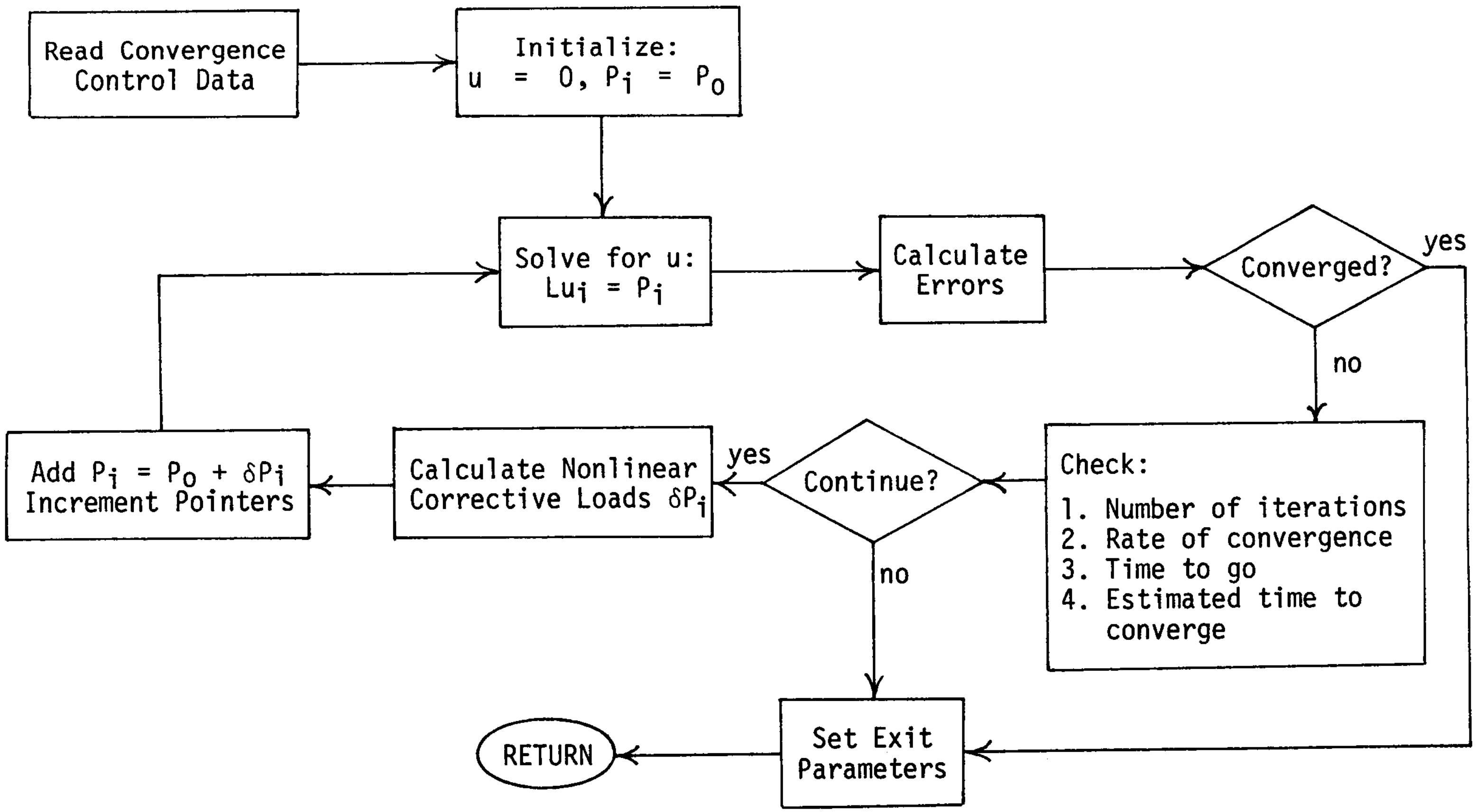

Figure 6.- Logical flow for module CAITER.