NASA CONTRACTOR REPORT

NASA CR-2568

NASA CR

PLANS - A FINITE ELEMENT PROGRAM FOR NONLINEAR ANALYSIS OF STRUCTURES

Volume I - Theoretical Manual

A. Pifko, H. S. Levine, and H. Armen, Jr.

Prepared by
GRUMMAN AEROSPACE CORPORATION
Bethpage, N.Y. 11714
for Langley Research Center

NATIONAL AERONAUTICS AND SPACE ADMINISTRATION • WASHINGTON, D. C. • NOVEMBER 1975

1. Report No. NASA CR-2568	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle PLANS - A Finite Element Program for Nonlinear Analysis of Structures Volume I — Theoretical Manual		5. Report Date November 1975
		6. Performing Organization Code
7. Author(s) A. Pifko, H. S. Levine, and H. Armen, Jr.		8. Performing Organization Report No. RE-501
		10. Work Unit No.
9. Performing Organization Name and Address Grumman Aerospace Corporation Research Department Bethpage, New York 11714		11. Contract or Grant No. NAS 1-10087
		13. Type of Report and Period Covered Final Report
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546		14. Sponsoring Agency Code
15. Supplementary Notes This is a topical report.		

16. Abstract

This report describes the PLANS system, a finite element program for nonlinear analysis that was developed at Grumman Aerospace Corporation under partial NASA sponsorship. Rather than being a single comprehensive computer program, PLANS represents a collection of special purpose computer programs each associated with a distinct physical problem class. Modules of PLANS specifically referenced and described in detail include: 1) REVBY, for the plastic analysis of bodies of revolution; 2) OUT-OF-PLANE, for the plastic analysis of 3-D built-up structures where membrane effects are predominant; 3) BEND, for the plastic analysis of built-up structures where bending and membrane effects are significant; 4) HEX, for the 3-D elastic-plastic analysis of general solids; and 5) OUT-OF-PLANE-MG, for material and geometrically nonlinear analysis of built-up structures. In addition, the SATELLITE program for data debugging and plotting of input geometries is also described.

This volume presents the theoretical foundations upon which the analysis is based. Discussed are the form of the governing equations, the methods of solution, plasticity theories available, a general system description and flow of the programs, and the elements available for use.

17. Key Words (Suggested by Author(s)) Finite Element, Plastic, Large Deflection, Nonlinear		18. Distribution Statement Unclassified - Unlimited; Subject Category 39	
19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of Pages 179	22. Price* $7.00

*For sale by the National Technical Information Service, Springfield, Virginia 22161

FOREWORD

The development of the PLANS (PLastic and large deflection ANalysis of Structures) system was conducted by the Grumman Aerospace Corporation, Bethpage, New York, under partial support of Contract NAS 1-10087, entitled "Nonlinear Analysis of Structures." The work was performed by the Research Department of Grumman Aerospace Corporation, with support from Grumman Data Systems.

Many people contributed in the development of the PLANS system. Thanks go to our colleague, Dr. Alvin Levy, for developing some of the element capability in PLANS and for his continuous interest and helpful comments. The authors acknowledge the contribution of Joseph S. Miller for his efforts associated with the initial programming of the PLANS system.

Special thanks go to Miss Patricia Zirk for her diligence and dedication to the programming effort required for the development of the PLANS system, and for seeing through the often thankless tasks involved in the development of a large program.

TABLE OF CONTENTS

Section		Page

Section			Page

Section			Page

LIST OF ILLUSTRATIONS

"PLANS — A Finite Element Program for Nonlinear Analysis of Structures, Volume I - Theoretical Manual"

by

A. Pifko, H. S. Levine, and H. Armen, Jr.

Grumman Aerospace Corporation
Bethpage, New York 11714

1. INTRODUCTION AND SUMMARY

This report describes the current state of the PLANS system, a finite element program for the nonlinear analysis of structures. PLANS, rather than being a single comprehensive computer program, represents a collection of special purpose computer programs or modules, each associated with a distinct physical problem class. Utilizing this concept, each module is an independent finite element computer program with its associated element library that can be individually loaded and used to solve the problem class of interest. In this manner, any simplification or specialization germane to the individual analysis can be incorporated within each program thereby optimizing computational efficiency. Thus, unavoidable inefficiencies that result from a more general purpose program are avoided in each special purpose module.

All the programs in PLANS employ the "initial strain" concept within an incremental procedure to account for the effect of plasticity and include the capability for cyclic plastic analysis. Geometric nonlinearities included in several of the modules are treated by using the "updated" or convected coordinate approach. These nonlinear terms may be incorporated as effective loads and/or modifications to the stiffness matrix.

The increased demands from governmental and industrial organizations associated with aerospace, naval, and nuclear reactor technology for determining accurate stress, strain, and displacement fields have been a motivating force behind the activity and advances in the development of techniques for the nonlinear analysis of structures.

As a result of these advances, the computational capability available for the nonlinear analysis of structures has experienced a tremendous growth during the past ten years. Indeed, the level of structural analysis capability that was achieved has outstripped our ability to describe accurately complex material behavior such as cyclic-, time-, and temperature-dependent plasticity. Prior to the development of the programs now available, the designer or analyst, confronted with a problem involving structural nonlinearities, was left with a choice of using his engineering judgment alone, or in conjunction with potentially expensive laboratory tests. He now has the further option of performing numerical analysis to gain insight into the behavior of the structure.

Most nonlinear analysis programs, with the exception of a few, have been developed as a spin-off of existing programs that were originally designed for linear structural analysis. Although this development is a natural one, the added dimension of generality has placed great responsibilities on the user of such programs. Perhaps the greatest asset of these programs, the ability to solve sophisticated problems, also represents a potential liability, i.e., they always produce numbers. The user must still exercise engineering judgment in order to interpret the results meaningfully. The analytic results will confirm these feelings and provide him with additional insight. However, he now has a luxury of having his intuition fail him without suffering the consequences of a catastrophic failure, or an overdesigned system.

The progress associated with solving nonlinear problems has been the direct result of advances in several interdisciplinary areas that include structural mechanics, numerical analysis, and computer systems engineering. Specifically, in the area of structural mechanics, significant advances have been those associated with the finite element method. The period of these advances for linear elastic behavior can be traced to the development of the displacement method of finite element analysis (Ref. 1).

References 2-16 are representative of advances associated with incorporating the effects of material nonlinear behavior into a finite element analysis. The approach taken in these references is to develop solutions of the repetitive type that linearize the basically nonlinear problem for a sequence of loading steps. These developed techniques fall into two categories: the initial strain (Refs. 2-10) and tangent modulus (Refs. 11-16) methods. They differ computationally depending on whether the effect of material nonlinearities enters as pseudoloads (initial strain) or the stiffness matrix is explicitly altered (tangent modulus).

Considerable effort has also been directed towards the treatment of geometric nonlinearities. These efforts are reported in Refs. 17-22 for consideration of geometric nonlinearities alone while the simultaneous treatment of both types of nonlinearities was reported in Refs. 23-27.

The concurrent contribution from numerical analysis methods notably has been the development of efficient algorithms to solve, on a repetitive basis, large systems of equations. The ease with which the theoretical considerations and numerical algorithms are efficiently processed and translated into meaningful answers is attributable to the ingenuity and talents of computer systems engineers who develop the necessary hardware and, in some cases, software to satisfy what must seem to them the insatiable appetite of the developers of large scale programs.

The advances in the above-mentioned areas have resulted in the development of many software systems to treat nonlinear behavior. These range from simple instructional programs to general comprehensive computer systems capable of treating the entire visible spectrum of two and three dimensional problems. Reference 28 presents in some detail the current capability of the various general purpose finite element programs that are available for plastic analysis while Ref. 29 details the available geometric nonlinear programs.

This report describes the theoretical basis of the PLANS system, a collection of special-purpose finite element computer programs for nonlinear analysis. These programs are an outgrowth of the work cited in Refs. 6, 8, and 30 conducted by the authors on the development and implementation of finite element methods for nonlinear analysis. As such, in order to give an historical prospective of the program's evolution, we first chronologically summarize the development of the capabilities represented by PLANS.

1.1 Chronology of the Development of Nonlinear Structural Analysis Techniques at Grumman Aerospace Corporation

- **Contract:** NAS 1-5040

  Period: 5/65-12/66

  Ref. Document: Ref. 6

  Goal: To combine existing finite element technology with plasticity theories to treat general (including cyclic) loading conditions.

  Result: Methodology developed for membrane stressed structures (two dimensional). Some verification with NASA experiments on notched panels under cyclic loading.

  Comments: The plastic analysis program developed at this stage required a separate elastic analysis program to generate influence coefficients. A restriction of a maximum of 55 elements with approximately 100 degrees of freedom was enforced on the basis of core storage limitations. This early effort is notable for its introduction of a reasonably sophisticated plasticity theory (kinematic hardening) within the framework of a complex numerical structural analysis procedure. The synthesis of this capability has served as a forerunner for subsequent efforts.

- Contract: NAS 1-7315

  Period: 6/67-6/69

  Ref. Document: Ref. 8

  Goals: 1) Extend methods developed in previous study to treat nonlinear bending behavior; and 2) determine the feasibility of treating combined material and geometric nonlinearities.

  Results: 1) Introduced and applied the concept of an elastic-plastic boundary in the plane and through the thickness of triangular and rectangular bending elements. Treated problems associated with bending alone or combined with membrane loads. 2) Incorporated, on a limited basis, an incremental technique to account for combined material and geometric nonlinearities.

  Comments: Applications to beams and plates. Determined collapse loads and modes of a variety of plate-type structures. Combined nonlinearities treated for beams and arches.

- Contract: NAS 1-10087

  Period: 6/70-6/74

Ref. Document: Ref. 30 and current report

Goals: 1) Expand "library" of finite elements to include a broad base of applications; 2) develop general comprehensive program for the plastic analysis of structures; 3) extend the methods for the treatment of combined material and geometric non-linearity; and 4) implement combined nonlinear analysis techniques into the program developed in (2).

Results: 1) In addition to triangular membrane and plate elements incorporated in the first two studies, the following elements were included into the element library: axi-symmetric revolved triangle, axisymmetric shell, 3-D variable node isoparametric solid element, shear panel, stringers, axisymmetric rings, general beams of various cross sections, and a composite element. 2) PLANS has evolved as a collection of programs each one of which is designed to treat a particular class of analysis, i.e., axisymmetric analysis of bodies of revolution, 3-D analysis of solid bodies, etc. (the particular modules are discussed in more detail in a subsequent section of this report).
3) General techniques to treat combined nonlinearities have been developed within the framework of the previous procedures

to treat plasticity alone. 4) The combined nonlinear analysis techniques were implemented into PLANS for the analysis of axisymmetric shell structures and for general 3-D built-up structures, typical of aircraft construction.

Comments: The successful completion of the goals associated with this effort has enabled analysts to realistically consider problem areas that could otherwise be only treated on a heuristic basis. Several of these problem areas are discussed subsequently in this section.

1.2 Applicable Problem Areas - Current and Future

Only two related motivating factors could justify the cost associated with developing advanced analytic capabilities such as those represented by PLANS. The first motivation is that the investment will result in reducing the cost associated with designing and/or building the system. The second reason for developing advanced analyses is that the system being considered does not fall into a general category of those previously designed and in service. Therefore, there must be the capability to gather insight by test and analysis to demonstrate the integrity of the system.

In addition to the problems associated with reducing weight and treating overload conditions, the following problem areas appear to be the most likely to benefit — from the viewpoint of the above-mentioned motivating factors — by the further development of nonlinear structural analyses.

1.2.1 Fatigue and Fracture Analysis

Fatigue life predictions are by-and-large semiempirical procedures. The techniques generally used have, until recently, paid little more than lip service to the effects of plastic flow. It is not realistic to consider such phenomena as the effect of residual stresses, propagating cracks, and cyclic load spectrum in a flawed structure without a reasonably accurate treatment of plastic deformation. Also included in the fatigue and fracture analysis problem area is fastener technology, where the installation and operation of the fasteners involve large strains and deformations.

1.2.2 Crashworthiness Evaluation

A capability for the analysis of general transportation systems is valuable for evaluating existing designs, post-mortem analysis of damaged units, predicting the crashworthiness of proposed designs, and establishing crashworthiness design criteria. Results of crash simulation studies can be used to determine probable damage to passengers, equipment, and structure. In addition, areas for structural modifications (including energy absorbers) can be determined along with their associated cost and/or weight penalties. Note that the application of such a capability is not limited to transportation systems, but include design and analysis of such stationary structures as highway barriers and nuclear containment vessels subjected to impact loads.

1.2.3 High Temperature Applications

In the design and analysis of nuclear reactor system components, large thermal gradients are common; in fact, much of the plastic deformation results from thermally induced loadings.

Furthermore, these loadings tend to be cyclical, separated by hold periods under sustained high levels of constant stress or strain conditions. The space shuttle represents another example of a structure that will operate in an environment in which temperatures are sufficiently high to cause degradation of structural strength. For both the reactor components and the space shuttle it becomes difficult to separate the effects associated with plastic flow and creep.

1.2.4 Metal Forming Technology

Ductile metals may be rolled, hammered, bent, or extruded even at relatively low temperatures. These processes generally require considerable energy. Therefore accurate methods for determining the behavior of solids, under the conditions severe enough to cause permanent changes in shape, are significant in attaining increased fabrication efficiency.

1.2.5 Nonisotropic Materials

The treatment of the nonlinear behavior of structures constructed of homogeneous orthotropic or layered composite materials may appear to be presumptuous in view of the many remaining unanswered questions associated with homogeneous isotropic materials. Nevertheless such materials exist (more so than the idealized materials considered) and are being used with increasing frequency in aerospace applications. It would be unrealistic to neglect the special problems posed by such materials when developing a general purpose plastic analysis capability.

2. PROGRAM DEFINITION

2.1 General Discussion

Developing a comprehensive finite element program remains a subjective undertaking since it invariably depends on the analysts designing the program, the computer hardware available, and the resources allocated for the project.

Many of the basic criteria for both linear and nonlinear general purpose codes are similar. However, given that the appropriate theories from structural mechanics have been implemented, the distinguishing features of a nonlinear program are twofold: 1) displacements, stresses, elastic-plastic strains, etc., must be stored for subsequent use in succeeding load steps, and 2) the solution is of the repetitive type so that calculations that are performed once for an elastic analysis must be repeated in a nonlinear analysis. The latter consideration is most critical because while it is true that nonlinear analyses have reached the stage of maturity such that almost any problem can be solved for a price, the cost for such an analysis can become prohibitive. Consequently, it is incumbent on the designer of a nonlinear code to minimize the cost of an analysis so that solutions to meaningful problems are economically feasible.

For a large comprehensive program, steps towards this end must be taken not only in reducing computing time (i.e., use of the central processing unit) but also in reducing overhead costs associated with accessing data on secondary storage devices.

The use of an efficient solution algorithm has the most significant effect in reducing computer costs associated with CPU time.

After the choice of a solution algorithm has been made, great care must be taken in coding the key "number crunching" portion of the code, including the judicious use of machine language routines for certain operations. Efforts must also be made to perform key input/output functions in an optimum fashion so that their associated overhead costs are minimized. The allocation between scratch disc storage and primary in-core storage is the main operational decision to be made here. Such features as variable core allocation enable this ratio to be optimized for a given problem and machine.

2.2 The PLANS System

The PLANS system, rather than being one comprehensive computer program, is actually a collection of finite element programs used for the nonlinear analysis of structures. This collection of programs evolved and is based on the organizational philosophy in which classes of analysis are treated individually based on the physical problem class to be analyzed. On the basis of this concept, each of the independent finite element computer programs of PLANS, with an associated element library can be individually loaded and used to solve the problem class of interest. A number of programs have been developed for material nonlinear behavior alone and for combined geometric and material nonlinear behavior. Table 1 summarizes the usage, capabilities, and element libraries of the current programs of the PLANS system. These include:

- REVBY — for the plastic analysis of bodies of revolution
- OUT-OF-PLANE — for the plastic analysis of built-up structures where membrane effects predominate

Table 1

CURRENT CAPABILITY OF PLANS

Module	Usage	Element Library	Loading	Current Capability
OUT-OF-PLANE (material nonlinearity)	Analysis of built-up structures composed of membrane skins, stringers, bulkheads	Family of triangular membrane elements, stringer element, shear panel, beam elements, with the following cross sections: solid rectangular, solid circular, I-section, Z-section, L-section, T-section, hollow rectangular, hollow circular	- Concentrated forces and moments at nodes - Distributed edge load on triangular element - Distributed lateral load on beam elements - Thermal loading	600 members 900 nodes 6 degrees of freedom per node
REVBY (material nonlinearity)	Analysis of bodies of revolution	Isoparametric shell of revolution element, revolved triangular element, ring stiffener of various cross sections	Axisymmetric - Line loading - Distributed load on shell elements - Distributed load on revolved triangles - Thermal loading	600 members 900 nodes 6 degrees of freedom per node
SATELLITE	Mesh generation, plot idealization, data checking	—	—	—
O-PLANE-MG (material and geometric nonlinearity)	Same as OUT-OF-PLANE but includes the effect of geometric nonlinear behavior	Same as OUT-OF-PLANE	Same as OUT-OF-PLANE	Same as OUT-OF-PLANE
HEX (material nonlinearity)	Analysis of three dimensional solids	Isoparametric family of hexahedra	- Concentrated forces at nodes - Distributed surface load on a face of a hexahedra - Thermal loading	2500 members 2500 nodes 3 degrees of freedom per node
BEND (material nonlinearity)	Analysis of built-up structures, includes bending of sheet material	Family of triangular membrane elements, stringer element, beam element (same cross sections as OUT-OF-PLANE), higher order triangular plate element (bending and membrane)	- Concentrated forces at nodes - Distributed edge loads on triangular plate and membrane elements - Distributed lateral load on plate and beam elements - Thermal load	Open core - variable number of members and nodes Up to 12 degrees of freedom per node

- BEND — for the plastic analysis of built-up structures where bending and membrane effects are significant
- HEX — for problems requiring a three dimensional elastic, plastic analysis
- OUT-OF-PLANE-MG — for the material and geometric nonlinear analysis of built-up structures

Supplementing these is a SATELLITE program for data debugging and plotting of input geometries.

Developing the programs in this manner afforded some distinct advantages, particularly in a research environment, since we were able to keep the individual programs as simple as possible and, therefore, easily understood and modifiable. Furthermore, in general, each program follows the same basic flow and each contains many common subroutines. Consequently, once the basic flow and supervisory subroutines have been established, development could proceed as parallel efforts with different programs being worked on at the same time.

It also simultaneously allowed for the development of spin-off modules for special purpose analysis. An example of this, reported in Ref. 31, is the FAST module for fracture analysis. Another special purpose module developed under this concept is SATELLITE which integrates the input part of the programs with FORTRAN mesh generation subprograms and input plotting routines.

Another implication of this approach also arose in the development of PLANS. With the development of each succeeding program, new and hopefully better programming techniques were realized. These improvements were implemented in succeeding programs so that continued growth in program capability, generality, and efficiency was achieved.

Although the programs can be individually loaded it is convenient to integrate them into a single system by making each module callable from an executive program.

A brief discussion of the significant features of several components of the system follows.

2.2.1 Executive Program

The executive program is a small calling program that utilizes the computer's system to form an overlay structure. However, for an IBM facility we make use of a PL/1 program called WIZARD (Ref. 32), written at Grumman Data Systems, that allows a user to store many program decks on a magnetic tape or direct access volume.

This system is comprised of three separate collections of subprograms. These are:

- A source update program to allow a user to selectively edit and compile a new source program
- An object update program that maintains updated files of compiled source programs
- A file select program that selectively loads programs from the object file for execution

While the file select program can accept commands to load individual programs, it can also load groups of programs by simply supplying a group name that has been previously defined. In addition, each group definition may contain other group names in its definition. This feature is particularly meaningful for our purposes since each program naturally defines a group, and programs common to all groups (such as the solution package) are naturally subgroups.

The modules of the PLANS system are also operational on CDC computers. However, since PL/1 is unavailable on many CDC facilities, the system described above cannot be used. Consequently, on CDC machines we have made use of an overlay approach to implement the executive system. Specifically, use is made of the NASTRAN linkage editor facility to implement this overlay procedure.

2.2.2 Basic Flow of the Analysis Programs

Modules were developed in the PLANS system that treat material nonlinear behavior and combined material and geometric nonlinear behavior.

The modules that analyze material nonlinearities alone implement the initial strain approach which does not require the stiffness matrix be updated at any step in the analysis so that the program organization will differ from that for combined geometric and material nonlinear behavior. First, we discuss in detail the basic flow for a typical module in PLANS that treats material nonlinear behavior alone using the initial strain approach.

Figure 1 is a schematic representation of the flow of each of the analysis programs that implement this approach. It should be pointed out that the program outlined below does not fulfill the criterion of being general purpose in the sense that it can be used to perform analysis of different physical phenomena (geometric and material nonlinear behavior) and implement various solution procedures (tangent modulus and initial strain approach).

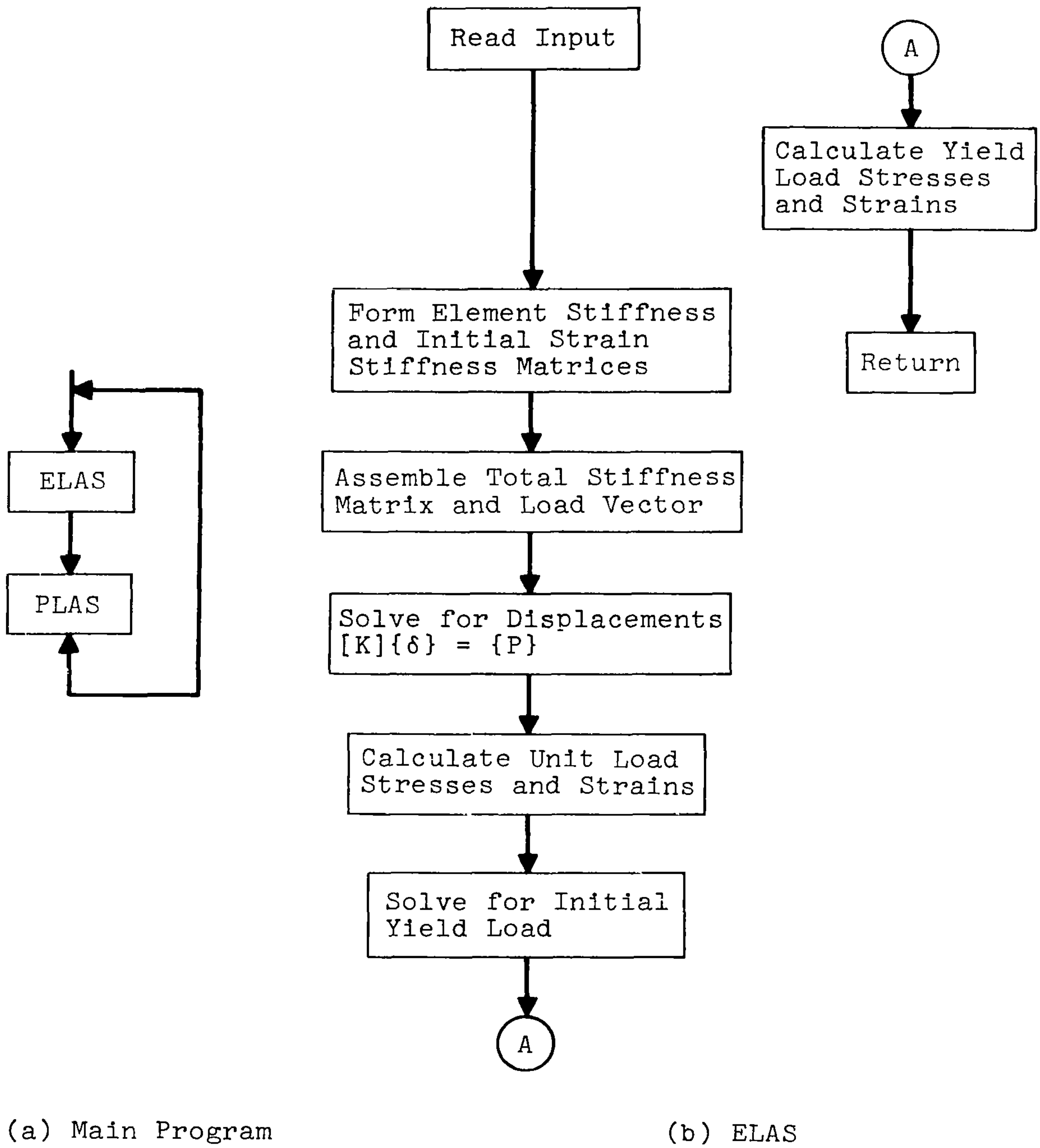


Fig. 1 Basic Flow of Nonlinear Material Analysis Program in PLANS

However, it was developed with the intent to implement one solution procedure for a given problem class in the simplest and most cost effective manner. In this sense the entire PLANS system and not an individual module can be considered to be a general purpose program.

As shown in Fig. 1, each module has three major components: a main calling routine (MAIN), an elastic analysis subprogram (ELAS), and the plastic analysis subprogram (PLAS). A discussion of each of these components follows.

- MAIN

Each individual computer program is controlled by a main calling program that sets up core allocation for principal arrays and the data set specifications for auxiliary storage. The main program then transfers control to subroutine ELAS. ELAS is a finite element elastic analysis program that performs the elastic analysis and calculates the initial yield load. Control is transferred back to the main program, which calls subroutine PLAS only if requested. PLAS manages the plastic analysis and maintains control of the analysis until the complete plastic analysis is performed. By so organizing the program it should be possible with a minimum amount of change to use PLAS with other available finite element elastic analysis programs.

- ELAS

Figure 1 shows a block diagram of the computational flow of ELAS. This program is a special purpose finite element program for the elastic analysis of structures. Accordingly, its first major task is to read all input. The input is read in functional groups as follows:

- Problem title
- Nodal coordinates and control variables
- Member topology describing connectivity
- Nodal boundary conditions. Single and multipoint constraints on the displacements are specified in addition to fixed or free conditions.
- Load vector – includes member or nodal load data for distributed surface loads, edge or line loads, concentrated loads, and thermal loads
- Material and section properties – tables of elastic and plastic material properties and member geometric properties are set up along with applicable members. Complete generality has been maintained so that orthotropic material behavior can be specified.

Other variables controlling output are also specified. The input scheme has been written so as to minimize the amount of card handling for any problem.

During the input portion of the program, externally numbered nodes and members are converted to internally numbered nodes and members via some internally generated tables. As a consequence of this feature, nodes and members can be arbitrarily numbered and new ones can be easily added to an existing idealization or the connectivity (bandwidth) altered with the specification of some simple card input. In addition, based on these tables, there is a substantial amount of input checking for the validity of specified nodes and members.

The next step is to form all element stiffness, stress, and initial strain stiffness matrices. This routine also places the elements of the stiffness matrix along with their position in the total stiffness matrix on a sequential auxiliary storage device. In addition, the element stress and initial strain matrix along with some element control variables are placed on another sequential auxiliary storage device to be used subsequently for stress-strain calculations.

The total stiffness matrix and load vector are assembled by sequentially reading the element stiffness matrix and "stacking" indices from auxiliary storage and stacking that portion of the stiffness matrix that fits in core, and then reading it onto an auxiliary storage device. This process is repeated until the entire load vector and stiffness matrix have been formed.

Our basic design philosophy is to make alterations, perform certain matrix manipulations such as those required for single and multipoint constraints (see Sec. 3.6), and account for boundary conditions (including applied displacements) on the element level, and then to assemble these quantities into the master arrays. In this manner we need not make use of a matrix interpretive system to manipulate large matrices.

With the total stiffness matrix and load vector assembled, the next step is to solve for displacements. Two solution packages are currently being used for this purpose, both based on the Cholesky factorization method. Their differences are based on the system used to store the stiffness matrix. PIRATE is based on partitioning the structure so that the stiffness matrix is explicitly in block tridiagonal form. The second routine, PODSYM, is based explicitly on the banded nature of the stiffness matrix, so that only elements within the lower triangle that lie between the semibandwidth need be assembled. In addition, this routine "packs" the rows of the stiffness matrix before writing it on auxiliary storage by suppressing all consecutive zeros. These algorithms are presented in detail in Sec. 3.7 of this manual.

The unit load stresses are calculated next from displacements. These stresses are used to calculate the initial yield load, and this yield load is used to scale the displacements and calculate yield load stresses and strains. Control is then returned to the main program.

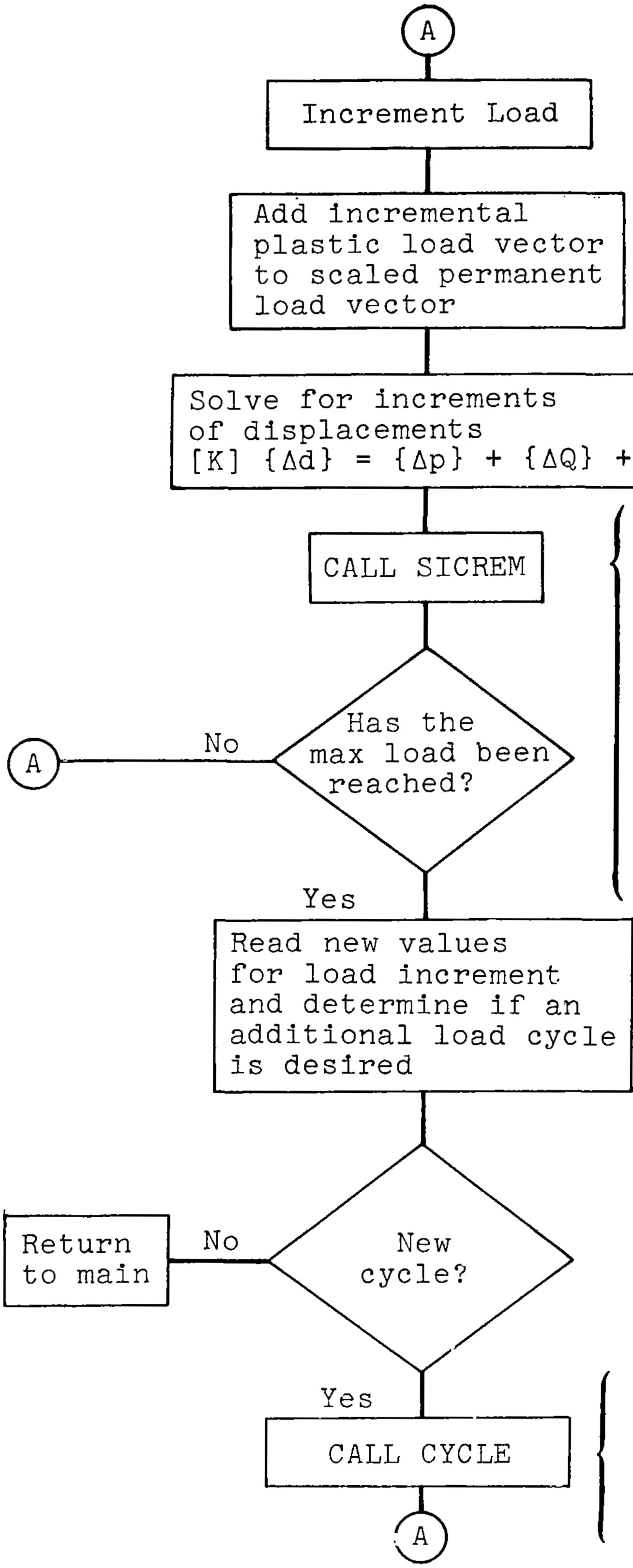
- PLAS

Figure 2 shows the computational flow of a typical subroutine PLAS. This program supervises the entire plastic analysis after initial processing has been carried out by ELAS. The principal information that is communicated to PLAS is:

- Factored stiffness matrix and unit load vector
- Element stress and initial strain matrices
- Initial yield load
- Plastic material properties

With this information, the load is incremented and the repetitive calculations that implement the incremental solution are performed as follows:

- Add incremental effective plastic load vector including any contribution from equilibrium correction factor to scaled vector of applied loads
- Solve for increments in displacements using the factored stiffness matrix. This involves only a forward and backward solution of a triangular matrix.



Subroutine SICREM supervises reading sequential auxiliary storage devices containing element stresses, total strains, etc. and calculates increments in stress, strain, and other pertinent quantities. The plasticity constitutive relations are implemented. The effective plastic load vector is formed, including the equilibrium correction factor. Total quantities are formed by summing incremental quantities. Print stresses, strains if requested.

Calculate "zero" load stresses and strains and displacements. Find new critical load and calculate new critical load stresses and strains.

Fig. 2 Flow of Subroutine PLAS

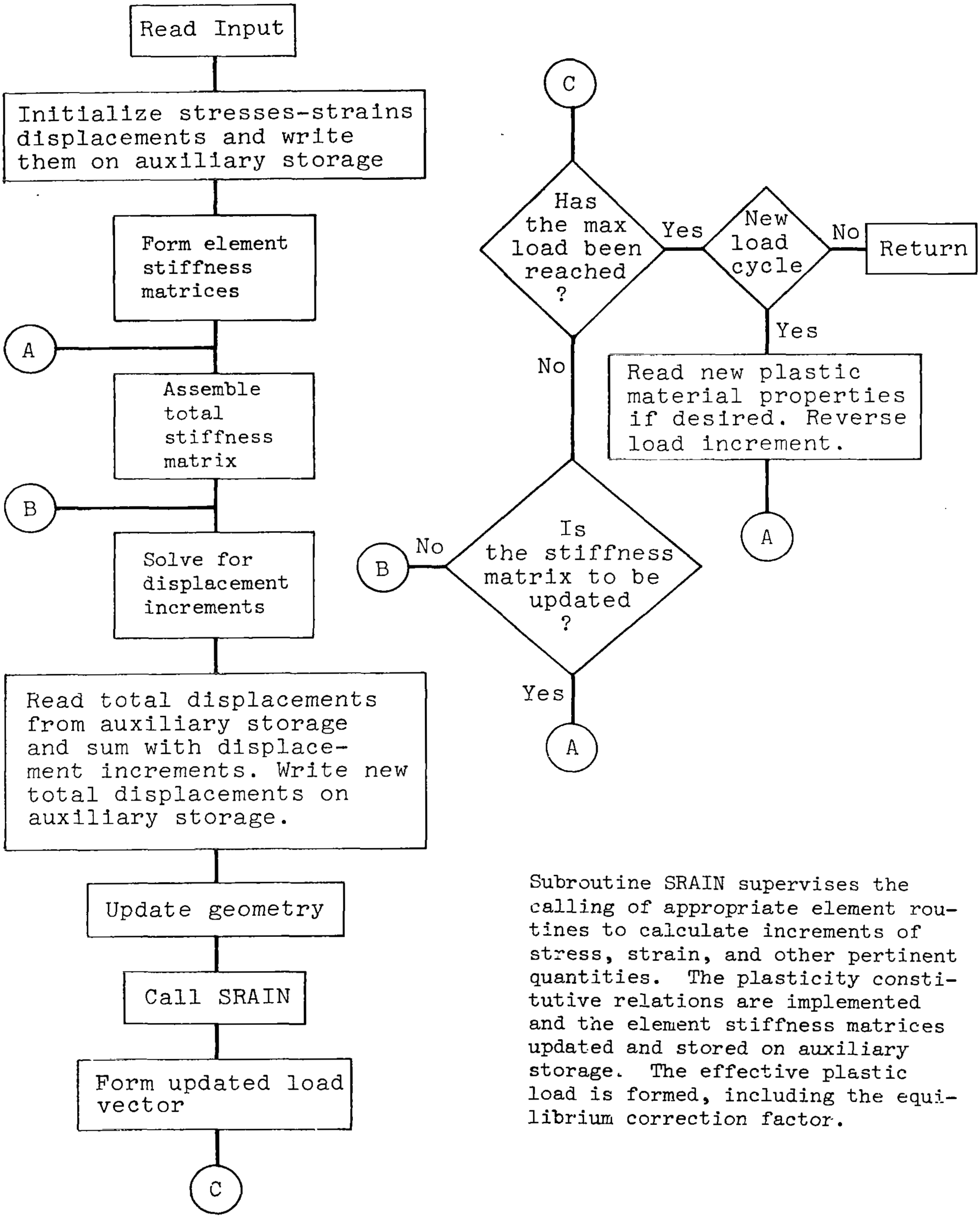
- Read total displacements from disc storage and add displacement increments to form updated total displacements. Write results on disc storage. Read total element (or nodal) stress, strains, plastic strains (shift in the yield surface when kinematic hardening is used) from disc storage. Calculate element (or nodal) increments of total strain, stress, etc., check yield criterion for previously elastic elements (nodes) and unloading condition for elements currently in the plastic range, implement plastic constitutive relations and form effective plastic load vector. Sum total quantities with calculated incremental quantities and write on disc storage to be used as input for the next increment of load.

- Repeat the above if the maximum load has not been reached.

When the maximum load is reached, the final effective plastic load vector is formed and saved. At this point, a check is made to see if another cycle of loading is desired. If not, control is returned to the main program. If an additional cycle of loading is specified, new plastic material parameters can be read as input and inserted into the material property tables. Displacements are then calculated with the applied load vector set equal to zero. Stresses and strains are calculated on the basis of these displacements. If subsequent yielding occurs in the reversed cycle at a load that is opposite in sign to the load previously computed, then these stresses and strains represent residual quantities. This is checked next by computing a new yield load. This calculation involves the solution of a quadratic equation for each member. One root of this solution represents the previously reached maximum load if no new plastic material properties were input, and the other represents the new yield load in the reversed direction. This load level is used as a starting point for the next cycle of loading. The effective plastic load vector is then added to the applied load vector, and the sum is used to solve for the new critical load displacements and the corresponding stresses and strains. Transfer is then made to the beginning of PLAS in order to increment the load and proceed as described previously.

2.3 Combined Material and Geometric Nonlinear Analysis

For combined material and geometric nonlinear analysis several major alterations in the program flow must be made because the convected coordinate approach to the solution of geometric nonlinear problems requires the reformation of the stiffness matrix during the incremental solution process. This flow is presented in Fig. 3. As shown in this illustration, no distinction can be made between an ELAS and a PLAS as before. The main program controls the flow previously supervised by ELAS and PLAS. Now there is a special routine to initialize stresses, strains, etc., whereas previously all quantities were initialized to be the yield load stresses, strains, and displacements. Aside from this distinction the flow of both analyses is similar. Now for the geometric nonlinear analysis, after an increment of load has been applied, increments of displacement are calculated and the geometry is updated. Subroutine SRAIN is then called. This subroutine supervises the elastic-plastic calculations. In addition to calculating the element stresses, strains, etc., using the appropriate stress-strain relations, the element stiffness matrices and mechanical load vector are updated because of the geometry changes and the



Subroutine SRAIN supervises the calling of appropriate element routines to calculate increments of stress, strain, and other pertinent quantities. The plasticity constitutive relations are implemented and the element stiffness matrices updated and stored on auxiliary storage. The effective plastic load is formed, including the equilibrium correction factor.

Fig. 3 Flow of Program for Combined Material and Geometric Nonlinear Behavior presence of initial stresses. These quantities are stored on auxiliary storage devices and control is returned to the main program. The new incremental load vector, including plastic and geometric effects as well as the "equilibrium correction," is then formed. If the maximum load has not been reached, control returns to point A or B in the flow depending upon whether the stiffness matrix is to be updated and refactored. If the maximum load for this load sequence has been reached new plastic material properties (and possibly a new load vector) are read and the new load increment is calculated. For reversed loading there can be no elastic scaling to the next yield load since the geometry changes make the unloading a nonlinear problem. Consequently the load sense is reversed, and the load is incremented to the new minimum load.

3. THEORETICAL APPROACH

This section describes the theoretical basis of the programs of the PLANS system. Included in this section are the formulation of the governing matrix equations that account for material and geometric nonlinear behavior, the description of the plasticity theory implemented in the programs, and some computational aspects of the programs. Much of the material in this section is explained in great detail in the authors' previous NASA contractor reports (Refs. 6, 8, and 30), and so the discussion in this section has been restricted to a review of the highlights. The above-mentioned reports may be referred to for additional details.

3.1 Governing Matrix Equations

The approach used by the authors in several investigations, spanning a number of years, for the plastic analysis of structures (Refs. 6, 8, 9, 10, 26, and 30), incorporates the initial strain concept for the treatment of material nonlinear behavior and the incremental moving coordinate formulation for the treatment of geometric nonlinear behavior. As such, this is the approach implemented in the programs of the PLANS system. An extensive presentation of the governing equations for this approach is presented in Ref. 8 and for completeness is reviewed below.

If the increments in displacements $\{\Delta U\}$, linear total strains $\{\Delta e_{ij}\}$, and rotations $\{\Delta\omega_{ij}\}$ are related to the nodal generalized displacements $\{\Delta U_o\}$ via the following matrix relations

$$
\begin{aligned}
\{\Delta U\} &= [N]\{\Delta U_o\} \\
\{\Delta e_{ij}\} &= [W]\{\Delta U_o\} \\
\{\Delta\omega_{ij}\} &= [\Omega]\{\Delta U_o\}
\end{aligned}
\tag{1}
$$

and Σ_{ij} is some initial stress state, then the matrix equation governing the nonlinear response of a structural finite element to some arbitrary increment of loading can be written as (Ref. 26)

$$([k^0] + [k^1])\{\Delta U_o\} = \{\Delta P_o\} + \{\Delta Q_o\} + \{R_o\} \tag{2}$$

where

$[k^0]$ = conventional stiffness matrix obtained from the linear component of the strain-displacement relations

$[k^1]$ = geometric stiffness matrix, formulated from the nonlinear terms of the strain-displacement relations. It can be considered as an additional component of the stiffness matrix that accounts for the effect that the presence of initial stresses have on subsequent deformations.

$\{\Delta P_o\}$ = increments of generalized nodal forces

$\{\Delta Q_o\}$ = increments of the effective plastic load. If an assumption is made on the plastic strain distribution within the element then $\{\Delta Q_o\} = [k^*]\{\Delta\epsilon_o\}$,

where

$[k^*]$ = initial strain stiffness matrix used to account for the presence of initial strains, and reflects the assumed distribution of both the total and plastic strains within the element

$\{\Delta\epsilon_o\}$ = increments of nodal or element plastic strain

$\{R_o\}$ = vector of residual forces that may exist because of equilibrium imbalance at the end of the previous load increment,

and

$$[k^0] = \int_V [W]^T[E][W]dV$$

$$[k^1] = \int_V [\Omega]^T[\Sigma_{ij}][\Omega]dV$$

$$\{\Delta Q_o\} = \int_V [W]^T[E]\{\Delta\epsilon_o\}dV$$

Equation (2) is derived by using a moving coordinate system fixed to the body. It is valid for large elastic-plastic deformations, provided the appropriate nonlinear terms are retained in the strain-displacement relations and the total strain increment can be simply decomposed into elastic and plastic components. Additionally, proper transformations from the previous to current coordinate systems must be used so that the changes in orientation and volume of the elements are accounted for (Refs. 26 and 24). In its usage herein, consideration is restricted to small strain, moderate rotation problems.

3.2 Solution Procedures

3.2.1 Small Deflection Problems (Material Nonlinearity Alone)

The matrix formulation presented in Eq. (2) is used for this case with the $[k^1]$ matrix deleted. The individual element stiffness matrices and load vectors are assembled to form the global stiffness matrix. Boundary conditions and single and multipoint constraint conditions are recognized at this stage and accounted for at the element level, thereby removing any further consideration of these conditions on the larger assembled coefficient matrices. At this stage we have

$$[k^0]\{\Delta U_o\}^{i+1} = \{\Delta P_o\}^{i+1} + \{\Delta Q_o\}^i + \{R_o\}^i \tag{3}$$

Here the superscripts i and i+1 refer to the current and next load step. Since the exact values of the plastic strain increments for the next load step are not known, we make use of values from the current step. The equilibrium correction is based on total equilibrium at the end of the current load step and is used in every increment. Note that the stiffness matrix need never be reformed after the first increment since the nonlinearities appear as a component to the load vector. Consequently, the stiffness matrix can be factored once for the initial load step and used repeatedly throughout the incremental solution. A significant reduction in solution time (factors of 3 to 21) have been realized depending on the size of the problem and the bandwidth using the previously factored coefficient matrix. The solution procedure is as follows.

A specified "unit" load is applied to the structure and corresponding stresses and strains are determined. Since the response is linear up to initial yield, a critical load for which plastic deformation first occurs at a node (or element) can be calculated from the unit load stresses. From this level the load is incremented to a maximum value with new increments of displacement, plastic strain and stress calculated at each step. Total values are obtained by summing incremental values.

If the load is reversed for a cyclic analysis, new material properties data may be read in at this point. A new critical load for which yielding begins in the reverse direction is calculated, based on elastic unloading to this point. Procedures for determining this load and initial yield load are presented in Refs. 6 and 8. The critical load for reversed loading may occur before all the initial load is removed because of the presence of residual stresses and the existence of the Bauschinger effect. The load is then incremented from the new critical value to a specified maximum (minimum) value. This procedure is repeated for as many half cycles as desired.

3.2.2 Combined Material and Geometric Nonlinearity

Again the formulation presented in Eq. (2) is employed. The load is applied in small increments from the initial unloaded state. At the end of each increment, new increments of deflection, stress, total strain, and plastic strain are calculated. Total quantities are summed and appropriate transformations from previous to current geometry are used to calculate such quantities as the initial stresses, Σ_{ij}. The geometry of the structure is updated in each step. Again the plastic strain increments used for the next step are those calculated at the end of the current step.

Now the element contributions are assembled and the system of linear incremental equations is solved for the next load increment. For the large deflection problem, the most time-consuming feature is the reassembly of the stiffness matrix in every increment, and the necessity of resolving the equations. To minimize this time-consuming feature it becomes convenient to treat the large deflection terms as effective loads and not reform the stiffness matrix $[k^0]$ in every increment (Ref. 26). We can then rewrite Eq. (2) as

$$[k^0]\{\Delta U_o\}^{i+1} = - [k^1]\{\Delta U_o\}^i + \{\Delta P_o\}^{i+1} + \{\Delta Q_o\}^i + \{R_o\}^i \tag{4}$$

Now the product of the geometric stiffness times the current displacement increment is treated as an "effective load." This is less accurate than the "tangent modulus" formulation but the "equilibrium correction" prevents excessive drifting of the solution. The stiffness matrix $[k^0]$ is updated every M steps ($M \geq 1$) where M is input by the user. Additionally, if large nonlinearities or instability are anticipated by the user, he may switch to the tangent modulus approach for geometric nonlinearity at any point in the analysis. After this specified load, the stiffness matrix $[k] = [k^0] + [k^1]$ is reformed and resolved for every increment of load.

The solution process is repeated until the maximum specified load is reached or structural failure occurs. If cyclic loading is specified, the load increment is reversed at the maximum load (a new magnitude may be input), new material properties data are read, as well as new values of M and the crossover load for tangent modulus treatment of geometric nonlinearities. The incremental process is then repeated until the new maximum (minimum) load is reached and the procedure is repeated for as many load cycles as desired.

3.3 Plasticity Relations

This section considers appropriate incremental plasticity relations to determine values of stress and plastic strain developed during the history of loading. In PLANS use is made of Hill's yield criterion (Ref. 33), for an orthotropic material, which reduces to the von Mises yield condition for isotropic materials, to predict initial yield and obtain the flow rules of plasticity. The capability of handling both strain hardening and ideally plastic behavior is included in the program.

3.3.1 Matrix Relations - Strain Hardening

We can, for small strain increments, decompose the total strain increment $\{\Delta e^T\}$ into elastic $\{\Delta e^e\}$ and plastic $\{\Delta \epsilon\}$ components, as

$$\{\Delta e^T\} = \{\Delta e^e\} + \{\Delta \epsilon\} \qquad (5)$$

The elastic strain increments are related to the stress increments $\{\Delta\sigma\}$ by

$$\{\Delta e^e\} = [E]^{-1}\{\Delta\sigma\} \tag{6}$$

where $[E]^{-1}$ is an array whose elements are combinations of elastic material constants.

A linear incremental constitutive relation between plastic strains and stresses can be written from all of the popularly used flow theories of plasticity. This relationship can be represented as

$$\{\Delta\epsilon\} = [C]\{\Delta\sigma\} \tag{7}$$

Therefore, substituting Eqs. (6) and (7) into Eq. (5) we can write

$$\{\Delta\sigma\} = [R]^{-1}\{\Delta e^T\} \tag{8}$$

where

$$[R] = [E]^{-1} + [C]$$

The elements of the matrix $[C]$ depend on the current state of stress, yield condition, flow rule, and hardening law. Examples of the explicit form of the $[C]$ matrix are given in Tables 2 and 3. In PLANS the yield condition is based on Hill's yield criterion for an orthotropic material (Ref. 33), and the hardening law is based on the Prager-Ziegler kinematic hardening theory (Refs. 34, 35, and 36) modified for orthotropic material behavior. Both linear and nonlinear strain hardening options are available with input parameters determining which is chosen. To minimize input requirements for nonlinear hardening, a Ramberg-Osgood (Ref. 37) representation of the stress-strain data is used

$$\epsilon = \frac{\sigma}{E} + \frac{3\sigma}{7E}\left(\frac{\sigma}{\sigma_{0.7}}\right)^{n-1} \tag{9}$$

Thus, for this representation of the stress-strain law, for an initially isotropic material two additional material parameters n and $\sigma_{0.7}$ are required for the plastic analysis. For a Ramberg-Osgood representation of an initially orthotropic material these two quantities are required for each principal material direction and for each of the shear components. For a three dimensional problem this means that six sets (three for the normal components and three for the shear components) of stress-strain data must be specified. The flow rule is based on Drucker's postulate (Ref. 38).

3.3.2 Matrix Relations - Perfect Plasticity

The treatment of multiaxial elastic ideally plastic behavior required that the following conditions be satisfied:

- The stress increment vector must be tangent to the loading surface
- The plastic strain increment vector must be normal to the loading surface, where the loading surface is the representation in stress space of the initial yield function or the subsequent yield function after some plastic deformation has occurred.

If $f(\sigma_{ij})$ represents the yield surface, the first condition can be expressed analytically as

$$\frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij} = 0 \quad , \tag{10}$$

Table 2 Isotropic $[C]$ Matrix for Various Stress States

PLANE STRESS

$$[C] = \frac{1}{D}\begin{bmatrix} m_1^2 & \text{symmetric} & \\ m_2 m_1 & m_2^2 & \\ m_3 m_1 & m_3 m_2 & m_3^2 \end{bmatrix} \qquad \begin{aligned} m_1 &= \bar{\sigma}_x - \tfrac{1}{2}\bar{\sigma}_y \\ m_2 &= \bar{\sigma}_y - \tfrac{1}{2}\bar{\sigma}_x \\ m_3 &= 3\bar{\tau}_{xy} \end{aligned}$$

$D = \frac{3}{2} c\sigma_o^2$, σ_o = yield stress , $\bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$

yield function: $f = \bar{\sigma}_x^2 + \bar{\sigma}_y^2 - \bar{\sigma}_x\bar{\sigma}_y + 3\bar{\tau}_{xy}^2 - \sigma_o^2 = 0$

unloading criterion: $m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\tau_{xy} < 0$

THREE DIMENSIONAL

$$[C] = \frac{1}{D}\begin{bmatrix} m_1^2 & & & \text{symmetric} & & \\ m_2 m_1 & m_2^2 & & & & \\ m_3 m_1 & m_3 m_2 & m_3^2 & & & \\ m_4 m_1 & m_4 m_2 & m_4 m_3 & m_4^2 & & \\ m_5 m_1 & m_5 m_2 & m_5 m_3 & m_5 m_4 & m_5^2 & \\ m_6 m_1 & m_6 m_2 & m_6 m_3 & m_6 m_4 & m_6 m_5 & m_6^2 \end{bmatrix} \qquad \begin{aligned} m_1 &= \bar{\sigma}_x - \tfrac{1}{2}(\bar{\sigma}_y + \bar{\sigma}_z) \\ m_2 &= \bar{\sigma}_y - \tfrac{1}{2}(\bar{\sigma}_z + \bar{\sigma}_x) \\ m_3 &= \bar{\sigma}_z - \tfrac{1}{2}(\bar{\sigma}_x + \bar{\sigma}_y) \\ m_4 &= 3\bar{\tau}_{xy} \\ m_5 &= 3\bar{\tau}_{xz} \\ m_6 &= 3\bar{\tau}_{yz} \end{aligned}$$

$D = \frac{3}{2} c\sigma_o^2$, σ_o = yield stress , $\bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$

yield function: $f = \frac{(\bar{\sigma}_y - \bar{\sigma}_z)^2}{2} + \frac{(\bar{\sigma}_z - \bar{\sigma}_x)^2}{2} + \frac{(\bar{\sigma}_x - \bar{\sigma}_y)^2}{2} + 3\bar{\tau}_{xy}^2 + 3\bar{\tau}_{xz}^2 + 3\bar{\tau}_{yz}^2 - \sigma_o^2 = 0$

unloading criterion: $m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\sigma_z + m_4 d\tau_{xy} + m_5 d\tau_{xz} + m_6 d\tau_{yz} < 0$

ONE NORMAL STRESS - TWO SHEAR COMPONENTS

$$[C] = \frac{1}{D}\begin{bmatrix} m_1^2 & \text{symmetric} & \\ m_2 m_1 & m_2^2 & \\ m_3 m_1 & m_3 m_2 & m_3^2 \end{bmatrix} \qquad \begin{aligned} m_1 &= \bar{\sigma}_x \\ m_2 &= 3\bar{\tau}_{xy} \\ m_3 &= 3\bar{\tau}_{xz} \end{aligned}$$

$D = \frac{3}{2} c\sigma_o^2$, σ_o = yield stress , $\bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$

yield function: $f = \bar{\sigma}_x^2 + 3\bar{\tau}_{xy}^2 + 3\bar{\tau}_{xz}^2 - \sigma_o^2 = 0$

unloading criteria: $m_1 d\sigma_x + m_2 d\tau_{xy} + m_3 d\tau_{xz} < 0$

Table 3 Orthotropic [C] Matrix for Various Stress States

PLANE STRESS

$$[C] = \frac{1}{D}\begin{bmatrix} m_1^2 & & \text{symmetric} \\ m_1 m_2 & m_2^2 & \\ m_1 m_3 & m_2 m_3 & m_3^2 \end{bmatrix} \qquad \begin{aligned} m_1 &= 2(G+H)\bar{\sigma}_x - 2H\bar{\sigma}_y \\ m_2 &= 2(F+H)\bar{\sigma}_y - 2H\bar{\sigma}_x \\ m_3 &= 4N\bar{\tau}_{xy} \\ m_4 &= -2F\bar{\sigma}_y - 2G\bar{\sigma}_x \end{aligned}$$

$$D = c\left(m_1^2 + m_2^2 + \frac{m_3^2}{2} + m_4^2\right) \ , \quad \bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$$

yield function: $f = (G+H)\bar{\sigma}_x^2 + (F+H)\bar{\sigma}_y^2 - 2H\bar{\sigma}_x\bar{\sigma}_y + 2N\bar{\tau}_{xy}^2 = 1$

unloading criterion: $m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\tau_{xy} < 0$

THREE DIMENSIONAL

$$[C] = \frac{1}{D}\begin{bmatrix} m_1^2 & & & & & \\ m_1 m_2 & m_2^2 & & \text{symmetric} & & \\ m_1 m_3 & m_2 m_3 & m_3^2 & & & \\ m_1 m_4 & m_2 m_4 & m_3 m_4 & m_4^2 & & \\ m_1 m_5 & m_2 m_5 & m_3 m_5 & m_4 m_5 & m_5^2 & \\ m_1 m_6 & m_2 m_6 & m_3 m_6 & m_4 m_6 & m_5 m_6 & m_6^2 \end{bmatrix} \qquad \begin{aligned} m_1 &= 2(G+H)\sigma_x - 2H\sigma_y - 2G\sigma_z \\ m_2 &= 2(F+H)\sigma_y - 2F\sigma_z - 2H\sigma_x \\ m_3 &= 2(F+G)\sigma_z - 2G\sigma_x - 2F\sigma_y \\ m_4 &= 4L\tau_{yz} \\ m_5 &= 4M\tau_{zx} \\ m_6 &= 4N\tau_{xy} \end{aligned}$$

$$D = c\left(m_1^2 + m_2^2 + m_3^2 + \frac{m_4^2}{2} + \frac{m_5^2}{2} + \frac{m_6^2}{2}\right) \ , \quad \bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$$

yield function: $f = F(\bar{\sigma}_y - \bar{\sigma}_z)^2 + G(\bar{\sigma}_z - \bar{\sigma}_x)^2 + H(\bar{\sigma}_x - \bar{\sigma}_y)^2 + 2L\bar{\tau}_{yz} + 2M\bar{\tau}_{zx} + 2N\bar{\tau}_{xy} = 1$

unloading criterion: $m_1 d\sigma_x + m_2 d\sigma_y + m_3 d\sigma_z + m_4 d\tau_{yz} + m_5 d\tau_{zx} + m_6 d\tau_{xy} < 0$

$$G+H = 1/X^2 \ , \quad H+F = 1/Y^2 \ , \quad F+G = 1/Z^2$$

$$2L = 1/R^2 \ , \quad 2M = 1/S^2 \ , \quad 2N = 1/T^2$$

X,Y,Z are yield stresses in tension in principal directions

R,S,T are yield stresses in shear in principal directions and provides a linear relationship among the components of stress increment. Thus, one of the components may be expressed in terms of the others. In matrix form, this can be written as

$$\{\Delta\sigma\} = [\underline{E}]\{\underline{\Delta\sigma}\} \tag{11}$$

where $\{\underline{\Delta\sigma}\}$ represents the independent stress components.

The normality condition provides a linear relation among the various components of the plastic strain increment. This condition is derived from the flow rule and provides a linear relationship in which each of the components of plastic-strain increment can be written in terms of any one component. This relationship may be represented in the following form

$$\{\Delta\epsilon\} = [\underset{\sim}{E}]\{\underset{\sim}{\Delta\epsilon}\} \tag{12}$$

where $\{\underset{\sim}{\Delta\epsilon}\}$ is the independent plastic strain increment.

The independent increments of stress and plastic strain can be combined and written as the components of a vector, $\{\Delta\omega\}$ (see Ref. 8), so that Eqs. (11) and (12) can be written, respectively, as

$$\begin{aligned} \{\Delta\sigma\} &= [\underline{E}]\{\Delta\omega\} \\ \{\Delta\epsilon\} &= [\underset{\sim}{E}]\{\Delta\omega\} \end{aligned} \tag{13}$$

Examples of the explicit form of $[\underline{E}]$ and $[\underset{\sim}{E}]$ are given in Tables 4 and 5. Combining the above equations with Eqs. (5) and (6), we can form the following relation for the independent quantities

$$\{\Delta\omega\} = [E^*]^{-1}\{\Delta e^T\} \tag{14}$$

where

$$[E^*] = [E]^{-1}[\underline{E}] + [\underset{\sim}{E}]$$

Table 4 Isotropic $[\bar{E}]$ and $[\tilde{E}]$ for Various Stress States

PLANE STRESS

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} , \quad [\tilde{E}] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = (\sigma_y - \tfrac{1}{2}\sigma_x) \big/ (\sigma_x - \tfrac{1}{2}\sigma_y)$$

$$m_2 = 3\tau_{xy} \big/ (\sigma_x - \tfrac{1}{2}\sigma_y)$$

THREE DIMENSIONAL

$$[\bar{E}] = \left[\begin{array}{c|ccccc} 0 & -m_2 & -m_3 & -m_4 & -m_5 & -m_6 \\ 0 & & & & & \\ 0 & & & & & \\ 0 & & & I & & \\ 0 & & & & & \\ 0 & & & & & \end{array}\right] , \quad [\tilde{E}] = \left[\begin{array}{c|ccccc} 1 & 0 & 0 & 0 & 0 & 0 \\ m_2 & & & & & \\ m_3 & & & & & \\ m_4 & & & 0 & & \\ m_5 & & & & & \\ m_6 & & & & & \end{array}\right]$$

$$m_1 = \sigma_x - \tfrac{1}{2}(\sigma_y + \sigma_z) , \quad m_2 = \left(\sigma_y - \tfrac{1}{2}(\sigma_x + \sigma_z)\right) \big/ m_1$$

$$m_3 = \left(\sigma_z - \tfrac{1}{2}(\sigma_x + \sigma_y)\right) \big/ m_1 , \quad m_4 = 3\tau_{xy}/m_1$$

$$m_5 = 3\tau_{xz}/m_1 , \quad m_6 = 3\tau_{yz}/m_1$$

ONE NORMAL STRESS - TWO SHEAR COMPONENTS

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} , \quad [\tilde{E}] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = 3\tau_{xy}/\sigma_x , \quad m_2 = 3\tau_{yz}/\sigma_x$$

Table 5 Orthotropic $[\bar{E}]$ and $[\tilde{E}]$ for Various Stress States

PLANE STRESS

$$[\bar{E}] = \begin{bmatrix} 0 & -m_1 & -m_2 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \quad , \quad [\tilde{E}] = \begin{bmatrix} 1 & 0 & 0 \\ m_1 & 0 & 0 \\ m_2 & 0 & 0 \end{bmatrix}$$

$$m_1 = \left((G+H)\sigma_y - H\sigma_x\right) \Big/ \left((F+H)\sigma_x - H\sigma_y\right)$$

$$m_2 = 2L\tau_{xy} \Big/ \left((F+H)\sigma_x - H\sigma_y\right)$$

THREE DIMENSIONAL

$$[\bar{E}] = \left[\begin{array}{c|ccccc} 0 & -m_2 & -m_3 & -m_4 & -m_5 & -m_6 \\ 0 & & & & & \\ 0 & & & & & \\ 0 & & & I & & \\ 0 & & & & & \\ 0 & & & & & \end{array}\right] \quad , \quad [\tilde{E}] = \left[\begin{array}{c|ccccc} 1 & 0 & 0 & 0 & 0 & 0 \\ m_2 & & & & & \\ m_3 & & & & & \\ m_4 & & & 0 & & \\ m_5 & & & & & \\ m_6 & & & & & \end{array}\right]$$

$$m_1 = (G+H)\sigma_x - H\sigma_y - G\sigma_z \quad ; \quad m_2 = \left((F+H)\sigma_y - H\sigma_x - F\sigma_z\right) \Big/ m_1$$

$$m_3 = \left((F+G)\sigma_z - G\sigma_x - F\sigma_y\right) \Big/ m_1 \quad ; \quad m_4 = 2L\tau_{xy}/m_1$$

$$m_5 = 2m\tau_{xz}/m_1 \quad ; \quad m_6 = 2N\tau_{yz}/m_1$$

$$G+H = 1/X^2 \quad , \quad H+F = 1/Y^2 \quad , \quad F+G = 1/Z^2$$

$$2L = 1/R^2 \quad , \quad 2M = 1/S^2 \quad , \quad 2N = 1/T^2$$

X,Y,Z are yield stresses in tension in principal directions

R,S,T are yield stresses in shear in principal directions

These relations are used with Hill's yield criterion for initially orthotropic behavior which reduces to the von Mises yield criterion for initially isotropic materials.

3.4 Hardening Coefficient

One of the unresolved questions in the use of the kinematic hardening theory of plasticity (Refs. 34, 35, and 36) is the definition of the hardening coefficient, c, (appearing in [C], Tables 2 and 3) under multiaxial stress states. This is especially true for problems involving nonlinear hardening under cyclic loading. Several definitions have been proposed by the current authors (Refs. 6 and 8) and successfully used for initially isotropic material behavior. An extension of this definition for initially orthotropic behavior based on Hill's yield criterion has been recently proposed (Ref. 39).

Before discussing the hardening coefficient, let us first rewrite the basic equations required for a plastic analysis using kinematic hardening.

First, we need a yield condition

$$f(\sigma_{ij}) = 0 \tag{15}$$

Hill's yield criterion for orthotropic materials (Ref. 33), which reduces to the von Mises yield condition for isotropic materials, is used in PLANS

$$f = F(\bar{\sigma}_y - \bar{\sigma}_z)^2 + G(\bar{\sigma}_z - \bar{\sigma}_x)^2 + H(\bar{\sigma}_x - \bar{\sigma}_y)^2 \tag{16}$$

$$+ 2L\bar{\tau}_{yz}^2 + 2M\bar{\tau}_{zx}^2 + 2N\bar{\tau}_{xy}^2 = 1$$

where

$$\frac{1}{X^2} = G+H \qquad 2L = \frac{1}{R^2}$$

$$\frac{1}{Y^2} = H+F \qquad 2M = \frac{1}{S^2}$$

$$\frac{1}{Z^2} = F+G \qquad 2N = \frac{1}{T^2}$$

Here $\bar{\sigma}_{ij} = \sigma_{ij} - \alpha_{ij}$ with α_{ij} being the components of the shift in the yield surface and X,Y,Z are the yield stresses in tension in the principal directions of orthotropy and R,S,T are the yield stresses in shear with respect to the principal axes. For initially isotropic materials this reduces to the familiar von Mises yield condition

$$f = \left(\frac{\bar{\sigma}_x - \bar{\sigma}_y}{2}\right)^2 + \left(\frac{\bar{\sigma}_y - \bar{\sigma}_z}{2}\right)^2 + \left(\frac{\bar{\sigma}_z - \bar{\sigma}_x}{2}\right)^2 + 3\bar{\tau}_{xy}^2 + 3\bar{\tau}_{yz}^2 + 3\bar{\tau}_{zx}^2 - \sigma_o^2 = 0 \tag{17}$$

with σ_o equal to the initial yield stress in tension.

Stability requirements for use of Hill's form of the yield surface, i.e., ensuring that it remains a "closed ellipsoidal" surface, require that the following criteria must also be satisfied by the yield stresses (Ref. 40)

$$\begin{aligned} F_{11}F_{22} - F_{12}^2 &> 0 \\ F_{22}F_{33} - F_{23}^2 &> 0 \\ F_{33}F_{11} - F_{31}^2 &> 0 \end{aligned} \tag{18}$$

where

$$F_{11} = \frac{1}{X^2} \quad ; \quad F_{22} = \frac{1}{Y^2} \quad ; \quad F_{33} = \frac{1}{Z^2}$$

$$F_{12} = -\tfrac{1}{2}\left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)$$

$$F_{31} = -\tfrac{1}{2}\left(\frac{1}{Z^2} + \frac{1}{X^2} - \frac{1}{Y^2}\right)$$

$$F_{23} = -\tfrac{1}{2}\left(\frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2}\right)$$

If Eq. (18) is not satisfied by the yield stresses then some other yield criterion must be chosen (see Ref. 40, for example) and implemented in the code. This occurs frequently for composite materials.

The flow rule used in PLANS is based on Drucker's postulate (Ref. 38)

$$d\epsilon_{ij}^{P} = d\lambda \frac{\partial f}{\partial \sigma_{ij}} \tag{19}$$

where $d\lambda$ is a positive scalar quantity. For kinematic hardening it is given by

$$d\lambda = \frac{1}{c} \frac{\dfrac{\partial f}{\partial \sigma_{mn}} d\sigma_{mn}}{\left(\dfrac{\partial f}{\partial \sigma_{kl}}\right)\left(\dfrac{\partial f}{\partial \sigma_{kl}}\right)} \tag{20}$$

and c is the hardening coefficient to be defined. The incremental shift in the yield surface $d\alpha_{ij}$ is assumed to be along a radial line through the center of the yield surface as postulated by Ziegler (Ref. 36)

$$d\alpha_{ij} = d\mu(\sigma_{ij} - \alpha_{ij}) \tag{21}$$

and

$$d\mu = \frac{\frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij}}{(\sigma_{ij} - \alpha_{ij}) \frac{\partial f}{\partial \sigma_{ij}}} \tag{22}$$

In evaluating the terms of Eqs. (19) through (22) for subsets of the nine dimensional stress space, all terms should be retained in the yield function. Symmetry and setting stresses identically to zero should be done after the expressions have been obtained. The resulting expressions are termed complete kinematic hardening.

3.4.1 Kinematic Hardening Coefficient for Initially Isotropic Materials

If we evaluate c for a uniaxial state of stress and restrict ourselves to initially isotropic materials, using Eqs. (19), (20), and (17), we get

$$\frac{1}{c} = \frac{3}{2} \frac{d\epsilon^P}{d\sigma} \tag{23}$$

If we generalize to a multiaxial state of stress, we may consider σ and ϵ^P to be effective quantities $\tilde{\sigma}$ and $\tilde{\epsilon}^P$ [Eq. (9)] where for an isotropic material

$$\tilde{\sigma}^2 = \left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \left(\frac{\sigma_y - \sigma_z}{2}\right)^2 + \left(\frac{\sigma_z - \sigma_x}{2}\right)^2 + 3\tau_{yz}^2 + 3\tau_{zx}^2 + 3\tau_{xy}^2 \tag{24}$$

and

$$d\tilde{\epsilon}^P = \sqrt{2/3 \; d\epsilon_{ij}^P d\epsilon_{ij}^P}$$

Note the difference in definition of the "effective stresses," $\tilde{\sigma}$ and the yield function of Eq. (17).

If we use a Ramberg-Osgood representation of the stress-strain curve and consider the nonlinear term to represent the plastic strain contribution, then

$$\frac{1}{c} = \frac{3}{2}\frac{d\tilde{\epsilon}^{P}}{d\tilde{\sigma}} = \frac{3}{2}\left(\frac{3n}{7E}\left|\frac{\tilde{\sigma}}{\sigma_{0.7}}\right|^{n-1}\right) \tag{25}$$

or for linear strain hardening

$$\frac{1}{c} = \frac{3}{2}\frac{d\tilde{\epsilon}^{P}}{d\tilde{\sigma}} = \frac{3}{2}\left[\frac{1}{E}\frac{\left(1 - \frac{E_T}{E}\right)}{E_T/E}\right] \tag{26}$$

where E_T is the tangent modulus.

Various methods of choosing an appropriate value for E_T/E from a stress-strain curve have been suggested (Ref. 41, for example). The use of linear strain hardening can lead to substantial underpredication of plastic strains, in the range of strain where the material does not have a linear stress-plastic strain relationship. In turn, this can lead to serious discrepancies if the response to cyclic loading is desired.

To determine the values of n and $\sigma_{0.7}$ that best fit the actual stress-strain data, the method suggested by Ramberg-Osgood can be used if the strain range is sufficiently small. For this case

$$n = 1 + \frac{\log(17/7)}{\log(\sigma_{0.7}/\sigma_{0.85})}$$

The quantities $\sigma_{0.7}$ and $\sigma_{0.85}$ are the stresses at which the curve has secant moduli of $0.7E$ and $0.85E$, respectively.

If the strain of interest is sufficiently large so that the parameters as determined by the preceding process do not fit the curve well, then a power law representation to fit the actual data can be used

$$\epsilon = \frac{\sigma}{E} + \beta\sigma^{n} \tag{27}$$

Now having determined β and n to "best fit" the experimental data, the value of n input is that calculated, and the value of $\sigma_{0.7}$ input is derived by equating the nonlinear terms of Eqs. (27) and (9), i.e.,

$$\sigma_{0.7} = \left(\frac{3}{7E\beta}\right)^{\frac{1}{n-1}}$$

For cyclic loading the problem of how to determine c for load reversals subsequent to the initial loading is even more vexing especially for nonlinear hardening. For linear hardening a procedure for selecting the value of c and new yield stresses for succeeding cycles is suggested in Ref. 41. For nonlinear hardening it is desirable to reproduce the actual hysteretic stress-strain curves obtained from test data. Morrow (Ref. 42) details extensive cyclic tests on isotropic metals to determine the cyclic and hysteresis stress-strain curves. He has found that both the cyclic and hysteresis curves may be well represented by a power law similar to the Ramberg-Osgood curves. Based on his findings, we have incorporated into PLANS a hardening coefficient for load reversals following the initial monotonic loading defined as

$$\frac{1}{c} = \frac{3}{2}\frac{d\tilde{\epsilon}^P}{d\tilde{\sigma}} = \frac{3}{2}\left[\frac{3n}{7E}\left|\frac{\tilde{\sigma}}{2\sigma_{0.7}}\right|^{n-1}\right] \tag{28}$$

where we now define $\tilde{\sigma}$ to be

$$\tilde{\sigma}^2 = \left(\frac{\overset{*}{\sigma}_x - \overset{*}{\sigma}_y}{2}\right)^2 + \left(\frac{\overset{*}{\sigma}_y - \overset{*}{\sigma}_z}{2}\right)^2 + \left(\frac{\overset{*}{\sigma}_z - \overset{*}{\sigma}_x}{2}\right)^2 + 3\tau_{xy}^{*2} + 3\tau_{yz}^{*2} + 3\tau_{zx}^{*2}$$

where $\overset{*}{\sigma}_{ij} = \sigma_{ij} - \hat{\sigma}_{ij}$, and $\hat{\sigma}_{ij}$ is the last value of stress at the end of the previous loading range.

3.4.2 Orthotropic Hardening Coefficient

A generalization of the definition of c for initially orthotropic materials based on Hill's yield criterion is presented in Ref. 39. The basic highlights will be reproduced here.

We define c to be

$$\begin{aligned} \frac{1}{c} = {} & \frac{1}{c_x}\frac{\frac{\partial f}{\partial \sigma_x}\bar{\sigma}_x}{2f} + \frac{1}{c_y}\frac{\frac{\partial f}{\partial \sigma_y}\bar{\sigma}_y}{2f} + \frac{1}{c_z}\frac{\frac{\partial f}{\partial \sigma_z}\bar{\sigma}_z}{2f} \\ & + \frac{1}{c_{xy}}\frac{\frac{\partial f}{\partial \tau_{xy}}\bar{\tau}_{xy}}{2f} + \frac{1}{c_{yz}}\frac{\frac{\partial f}{\partial \tau_{yz}}\bar{\tau}_{yz}}{2f} + \frac{1}{c_{zx}}\frac{\frac{\partial f}{\partial \tau_{zx}}\bar{\tau}_{zx}}{2f} \end{aligned} \tag{29}$$

where

$$c_x = \frac{(G+H)^2}{\left[(G+H)^2 + H^2 + G^2\right]}\frac{d\sigma_x}{d\epsilon_x^P}$$

$$c_y = \frac{(F+H)^2}{\left[(F+H)^2 + H^2 + F^2\right]}\frac{d\sigma_y}{d\epsilon_y^P}$$

$$c_z = \frac{(F+G)^2}{\left[(F+G)^2 + F^2 + G^2\right]} \frac{d\sigma_z}{d\epsilon_z^P}$$

$$c_{xy} = 2\,\frac{d\tau_{xy}}{d\gamma_{xy}^P} \quad ; \quad c_{yz} = 2\,\frac{d\tau_{yz}}{d\gamma_{yz}^P} \quad ; \quad c_{zx} = 2\,\frac{d\tau_{zx}}{d\gamma_{zx}^P}$$

This form reduces to the individual uniaxial stress-strain laws and the initially isotropic case given in Eq. (23).

For a linear strain hardening approximation we use

$$\begin{aligned}
\frac{d\epsilon_x^P}{d\sigma_x} &= \frac{1}{E_x}\,\frac{\left(1 - E_{T_x}/E_x\right)}{\left(E_{T_x}/E_x\right)} \quad ; \quad \frac{d\epsilon_y^P}{d\sigma_y} = \frac{1}{E_y}\,\frac{\left(1 - E_{T_y}/E_y\right)}{\left(E_{T_y}/E_y\right)} \\
\frac{d\epsilon_z^P}{d\sigma_z} &= \frac{1}{E_z}\,\frac{\left(1 - E_{T_z}/E_z\right)}{\left(E_{T_z}/E_z\right)} \quad ; \quad \frac{d\gamma_{xy}^P}{d\tau_{xy}} = \frac{1}{G_{xy}}\,\frac{\left(1 - G_{T_{xy}}/G_{xy}\right)}{\left(G_{T_{xy}}/G_{xy}\right)} \qquad (30) \\
\frac{d\gamma_{yz}^P}{d\tau_{yz}} &= \frac{1}{G_{yz}}\,\frac{\left(1 - G_{T_{yz}}/G_{yz}\right)}{\left(G_{T_{yz}}/G_{yz}\right)} \quad ; \quad \frac{d\gamma_{zx}^P}{d\tau_{zx}} = \frac{1}{G_{zx}}\,\frac{\left(1 - G_{T_{zx}}/G_{zx}\right)}{\left(G_{T_{zx}}/G_{zx}\right)}
\end{aligned}$$

where E_{T_x}, E_{T_y}, etc., are specified by the user for each component direction. For a Ramberg-Osgood form of the uniaxial stress-strain data we use

$$\begin{aligned}
\frac{d\epsilon_x^P}{d\sigma_x} &= \frac{3n_x}{7E_x}\left[\frac{\sqrt{2/3\,\frac{(F+G+H)}{G+H}\,\tilde{\sigma}^2}}{\sigma_{0.7_x}}\right]^{n_x-1} \\
\frac{d\epsilon_y^P}{d\sigma_y} &= \frac{3n_y}{7E_y}\left[\frac{\sqrt{2/3\,\frac{(F+G+H)}{F+H}\,\tilde{\sigma}^2}}{\sigma_{0.7_y}}\right]^{n_y-1}
\end{aligned} \qquad (31)$$

$$\frac{d\epsilon_z^P}{d\sigma_z} = \frac{3n_z}{7E_z}\left[\frac{\sqrt{2/3\ \frac{(F+G+H)}{F+G}\ \tilde{\sigma}^2}}{\tau_{0.7_z}}\right]^{n_z-1}$$

$$\frac{d\gamma_{xy}^P}{d\tau_{xy}} = \frac{3n_{xy}}{7G_{xy}}\left[\frac{\sqrt{2/3\ \frac{(F+G+H)}{2N}\ \tilde{\sigma}^2}}{\tau_{0.7_{xy}}}\right]^{n_{xy}-1}$$

(31)
(Cont.)

$$\frac{d\gamma_{yz}^P}{d\tau_{yz}} = \frac{3n_{yz}}{7G_{yz}}\left[\frac{\sqrt{2/3\ \frac{(F+G+H)}{2L}\ \tilde{\sigma}^2}}{\tau_{0.7_{yz}}}\right]^{n_{yz}-1}$$

$$\frac{d\gamma_{zx}^P}{d\tau_{zx}} = \frac{3n_{zx}}{7G_{zx}}\left[\frac{\sqrt{2/3\ \frac{(F+G+H)}{2M}\ \tilde{\sigma}^2}}{\tau_{0.7_{zx}}}\right]^{n_{zx}-1}$$

where

$$\tilde{\sigma}^2 = \frac{3}{2(F+G+H)}\Big[F(\sigma_y - \sigma_z)^2 + G(\sigma_z - \sigma_x)^2 + H(\sigma_x - \sigma_y)^2 + 2L\tau_{yz}^2 + 2M\tau_{zx}^2 + 2N\tau_{xy}^2\Big] \qquad (32)$$

is the effective stress for initially orthotropic materials. Now n_x, $\sigma_{0.7_x}$, n_y, $\sigma_{0.7_y}$, etc., are specified by the user for each component direction.

For cyclic loading, we define $\tilde{\sigma}$ to be

$$\tilde{\sigma}^2 = \frac{3}{2(F+G+H)}\Big[F(\sigma_y^* - \sigma_z^*)^2 + G(\sigma_z^* - \sigma_x^*)^2 + H(\sigma_x^* - \sigma_y^*)^2 + 2L\tau_{yz}^{*2} + 2M\tau_{zx}^{*2} + 2N\tau_{xy}^{*2}\Big] \qquad (33)$$

where

$$\overset{*}{\sigma}_{ij} = \sigma_{ij} - \hat{\sigma}_{ij} \qquad (34)$$

Data for orthotropic cyclic loading is seriously lacking. If the representation given by Eqs. (33) and (34) for the effective stress is deemed valid then $\sigma_{0.7_{ij}}$ should be set to twice the measured monotonic value, for load reversals after the initial loading range. Additionally, at the end of each half-cycle of loading new plastic material properties may be input so that each hysteresis loop may be accurately represented.

3.5 Unloading Criterion

At times, due to nonproportional loading or other reasons, local unloading occurs even though the applied load is increasing. The unloading criterion, which is checked in the program for every load increment, is given by

$$\frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij} \geq 0 \qquad \text{for loading or neutral loading}$$

$$\frac{\partial f}{\partial \sigma_{ij}} d\sigma_{ij} < 0 \qquad \text{for unloading}$$

The values of $d\sigma_{ij}$ used in this calculation are obtained from the "elastic" stress-strain relations for that increment. They are the actual $d\sigma_{ij}$ if unloading is detected. If the unloading criterion is not met, however, the $d\sigma_{ij}$ values are determined from the plasticity constitutive relations. When unloading is detected at a point, all further stress and strain increments are elastic until reloading is detected using an appropriate yield criterion.

Specific relations for the unloading condition are shown in Tables 2 and 4.

3.6 Single and Multipoint Constraints

It is the philosophy of all the programs of the PLANS system to perform all matrix operations on the element level rather than on the assembled global matrices. Thus, element stress and strain recovery, load vector calculation, and boundary and multipoint constraint operations are carried out with respect to small matrices or vectors that readily fit in primary storage. In this manner, with the exception of the solution of the global stiffness equations, matrix operations can be accomplished without the aid of a matrix package for large scale matrix operations.

Accordingly, single and multipoint constraints are satisfied on the element level before assembling the total stiffness matrix. This can be illustrated by first writing the element influence coefficient equation as

$$\{f\} = [k]\{\Delta_g\} - [k^*]\{\epsilon\} \tag{35}$$

where

- $\{f\}$ is the vector of nodal generalized forces
- $\{\Delta_g\}$ is the vector of nodal generalized global displacements
- $\{\epsilon\}$ is some initial strain state

and $[k]$, $[k^*]$ are the element stiffness and initial strain matrices.

This equation can be partitioned into an analysis set that contributes to the total stiffness matrix and a reaction set that contributes only to the calculation of nodal reaction forces.

In PLANS, the reaction set for single point constraints arises from fixed node points or nodes that are given a prescribed nonzero generalized displacement component.

Accordingly, Eq. (35) becomes

$$\begin{Bmatrix} f_a \\ -- \\ f_r \end{Bmatrix} = \left[\begin{array}{c|c} k_{aa} & k_{ar} \\ \hline k_{ra} & k_{rr} \end{array}\right] \begin{Bmatrix} \Delta_g^a \\ -- \\ \Delta_g^r \end{Bmatrix} - \begin{bmatrix} k_a^* \\ -- \\ k_r^* \end{bmatrix} \{\epsilon\} \tag{36}$$

where

$$\{\hat{f}_a\} = [k_{aa}]\{\Delta_g^a\} \tag{37}$$

contribute to the total matrix equations with the load vector consisting of

$$\{\hat{f}_a\} = \{f_a\} - [k_{ar}]\{\Delta_g^r\} + [k_a^*]\{\epsilon\}$$

and the reaction forces are

$$\{f_r\} = [k_{ra}]\{\Delta_g^a\} + [k_{rr}]\{\Delta_g^r\} - [k_r^*]\{\epsilon\}$$

The above procedure is accomplished within the program without explicitly forming the partitioned matrix by making use of an element location vector that is formed when the boundary conditions are initially specified. This vector is of the form

$$\mathcal{L} = \begin{Bmatrix} 0 \\ n_i \\ n_j \\ 0 \\ -m_i \\ \vdots \end{Bmatrix} \tag{38}$$

where the zeros appear for fixed degrees of freedom, n_i, n_j, ... are associated with degrees of freedom that are free, and $-m_i$ indicates an applied displacement component.

The use of the element location vector is demonstrated symbolically by placing the vector as shown below

$$\begin{array}{c} \begin{bmatrix} 0 & n_i & n_j & -m_i & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix} \\ \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & \cdot & \cdot & \cdot & \cdot & \cdot \\ k_{21} & k_{22} & k_{23} & k_{24} & \cdot & \cdot & \cdot & \cdot & \cdot \\ k_{31} & k_{32} & k_{33} & k_{34} & \cdot & \cdot & \cdot & \cdot & \cdot \\ k_{41} & k_{42} & k_{43} & k_{44} & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \end{bmatrix} \begin{bmatrix} 0 \\ n_i \\ n_j \\ -m_i \\ \cdot \\ \cdot \end{bmatrix} \end{array}$$

Then all intersections of the 0, $-m_i$ rows and columns indicate matrix elements that are in k_{rr} and the intersection of 0, $-m_i$ with the n's indicates matrix elements that are in k_{ra} or k_{ar}. The intersection of the n's denotes elements in k_{aa}. The indices n_i, n_j denote the actual row-column location in the total stiffness matrix of the matrix element to be assembled. In practice however, the total stiffness matrix is assembled in a one dimensional array so that n_i, n_j are used to calculate a stacking index to relocate the matrix element in the one dimensional storage array.

The vector $\hat{f}_a$ is also assembled into the total load vector using $\mathcal{L}$. In this case all elements adjacent to 0, $-m_i$ are in f_r and those adjacent to n_i are in $\hat{f}_a$ and are, therefore, added to the "n_i"th location of the load vector.

The programs of the PLANS system implement a multipoint constraint capability of the form

$$\delta_i^d = \sum_{j=1}^{N} a_{ij}\delta_j \tag{39}$$

where δ_i^d is the i^{th} dependent degree of freedom, a_{ij} are prescribed coefficients, and δ_j are a set of independent degrees of freedom.

Equation (39) can be used to form a transformation matrix which contains the pertinent a_{ij} and maps the element displacement vector $\{\Delta_g\}$ which contains dependencies to a set containing only independent degrees of freedom.

Thus,

$$\{\Delta_g\} = [T]\{\tilde{\Delta}_g\} \tag{40}$$

from which Eq. (35) becomes

$$\{\tilde{f}\} = [\tilde{k}]\{\tilde{\Delta}_g\} - [\tilde{k}^*]\{\epsilon\} \tag{41}$$

where

$$\{\tilde{f}\} = [T]^T\{f\}$$

$$[\tilde{k}] = [T]^T[k][T]$$

$$[\tilde{k}^*] = [T]^T[k^*]$$

Again the procedure is accomplished without explicitly forming the transformation matrices or $\{\tilde{f}\}$, $[\tilde{k}]$, $[\tilde{k}^*]$. The location vector is used as before in conjunction with a dependency vector and three tables as shown in Fig. 4.

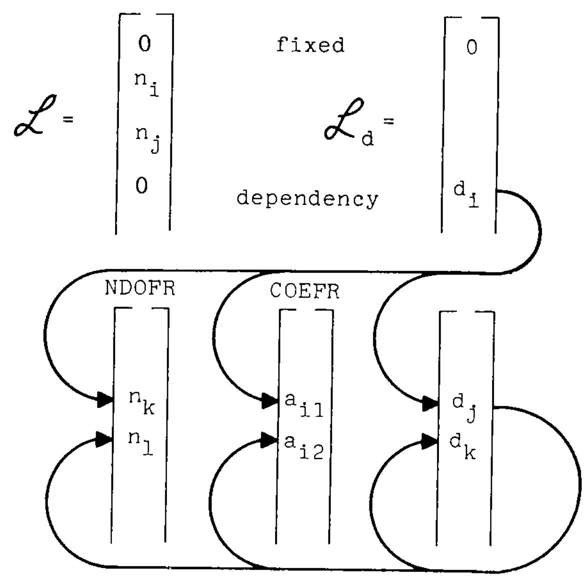


Fig. 4 Assembling with Multipoint Constraints

The dependency vector contains zero entries for all independent displacement components. All dependent displacement components are indicated by a zero in the appropriate location in $\mathcal{L}$. The index in $\mathcal{L}_d$ points to the "d_i"th location of three tables. The first gives the degree of freedom of the first independent displacement component in Eq. (39) and the second gives its associated coefficient. With this information, the matrix elements in [k] associated with the dependent displacements are appropriately multiplied by a_{ij} and then assembled according to n_k in the same manner outlined for single point constraints. The last table contains an index that points to the location of the a_{ij} and n_i for the next independent pair in Eq. (39). In this case, the assembling operation continues as before. A zero entry indicates the end of this dependency relation.

3.7 Equation Solver

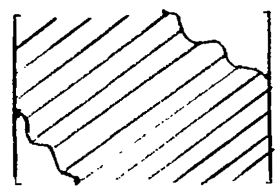
Two solution procedures are employed in the PLANS system for solution of the matrix equations. They are called PIRATE and PODSYM. PIRATE uses a solution algorithm based on the Cholesky decomposition scheme and relies on the fact that the matrices are positive definite and symmetric, and are explicitly in block tridiagonal form. Because the matrix must be in block tridiagonal form, for the most efficient storage allocation, the matrix must be partitioned. As a result, all the nodes in each partition must be specified and at any one time at least one diagonal and one off-diagonal block must be in core. Only the REVBY module of PLANS uses PIRATE because of the tight banding possible in most axisymmetric shell and body of revolution problems. In addition, at most, REVBY considers two dimensional meshes of elements which are relatively easy to partition. A detailed description of the PIRATE algorithm is presented in this section.

The second solution procedure, PODSYM, is used in the remaining modules of PLANS. This solution package requires the system of matrix equations to be banded, positive definite and symmetric. It, too, is based on the Cholesky algorithm but since this algorithm requires only one row of the matrix to be in core at any one time, no partitioning is required. Node order must be specified, however, and should be ordered so as to minimize the bandwidth and thus reduce solution time. A bandwidth optimizer is available in the SATELLITE program of PLANS for input data checking and plotting.

3.7.1 PODSYM - Solution of Symmetric Positive Definite Banded Matrix Equations

PODSYM solves the matrix equation $AX = Y$ where

A =



is a banded positive definite symmetric matrix, X is the desired solution vector, and Y is the known right side (load vector). PODSYM is the user interface and supervisory routine. It uses the Cholesky algorithm which factors the total stiffness matrix into LL^T (where L is a lower triangular matrix) and then solves a pair of triangular sets of equations.

The large positive definite matrices that occur in practical work very often contain a large number of zero entries and the program seeks to benefit from the presence of these elements by modifying the standard Cholesky formulas

$$\ell_{kk} = \left[A_{kk} - \sum_{j=1}^{k-1} \ell_{kj}^2 \right]^{\frac{1}{2}}$$

and

$$\ell_{ik} = \left[A_{ik} - \sum_{j=1}^{k-1} \ell_{ij}\ell_{kj} \right] / \ell_{kk} \qquad \text{for} \qquad i > k$$

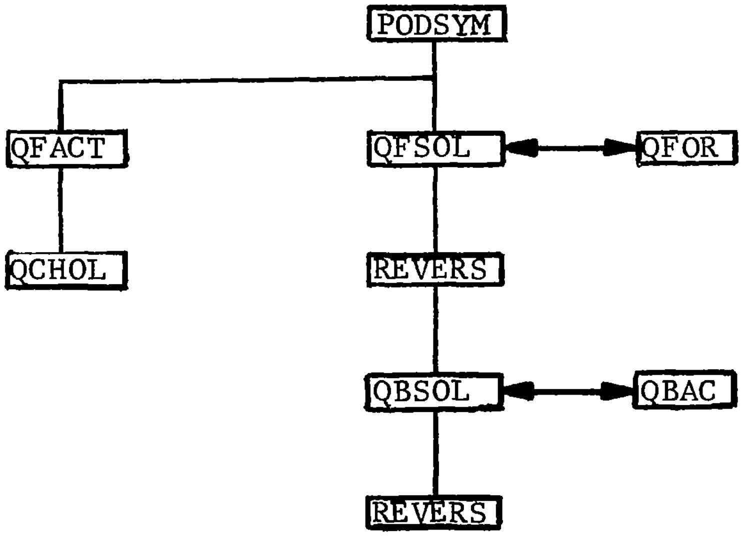
to read instead

$$\ell_{kk} = \left[A_{kk} - \sum_{j=\nu(k)}^{k-1} \ell_{kj}^{2} \right]^{\frac{1}{2}}$$

and

$$\ell_{ik} = \left[A_{ik} - \sum_{j=\mu(i,k)}^{k-1} \ell_{ij}\ell_{kj} \right] / \ell_{kk}$$

where $\mu(i,k)$ is the larger of the leading zeros in the i and k rows. In this manner multiplication by zero values of ℓ_{ij} is avoided. A flow chart for PODSYM is shown below



The factorization is carried out by subroutine QFACT which supervises the storage and data set allocation and subroutine

QCHOL which generates the lower triangular L matrix. QCHOL implements the Cholesky algorithm to factor the positive definite symmetric A matrix as the product of a lower triangular matrix with its transpose

$$A = LL^T$$

A straightforward argument establishes the possibility of decomposing any positive definite matrix in this fashion. Once L has been obtained, it is not difficult to solve the system of linear equations $AX = Y$ by calculating Z as the solution of the lower triangular system $LZ = Y$, and then X as the solution of the upper triangular system $L^T X = Z$. The forward solution $(LZ = Y)$ is accomplished by subroutines QFSOL and QFOR and the back solution $(L^T X = Z)$ by subroutines QBSOL and QBAC. Before the call to QBSOL, subroutine REVERS is called, which reverses the rows of L so that the last row becomes the first row, etc. This is accomplished in order to sequentially access L^T during the back solution.

The above algorithm is noteworthy for its assured stability and general efficiency. It is possible to carry out an error analysis of the procedure as it is represented on a digital computer. Such analysis shows that the computed L matrix satisfies an equation of the form

$$A + E = LL^T$$

with bounds on the elements of E which show that E is small compared to A. The effect of rounding errors made in the subsequent solution of $LZ = Y$ and $L^T X = Z$ may then be taken into account by (implicitly) introducing an additional perturbation F into A, and it is then concluded that the computed solution X_o exactly satisfies the equation

$$(A + E + F)X_o = Y$$

Since $E + F$ can be shown to be small, one would like to infer that X_o is almost an exact solution of the original equations, and unless A is too nearly singular, such a conclusion is indeed warranted. But if A is very ill-conditioned, no such result can be guaranteed, and X_o may be far from the mathematically correct solution; in this event single-precision computation will not suffice for the calculation of an accurate solution and since the solution will be very sensitive to small errors in A, it is unlikely that even a high-precision computation will yield satisfactory results unless A and Y are known (and supplied) to more than single-precision accuracy. The PODSYM subroutines make a fairly realistic attempt to detect and report pathological conditions of this sort.

3.7.2 PIRATE - Solution of Symmetric Positive Definite Matrix Equations in Block Tridiagonal Form

PIRATE solves the matrix equation $AX = Y$, where

$$A = \begin{bmatrix} A_1 & B_1 & & \\ B_1 & A_2 & B_2^T & \\ & B_2 & A_3 & B_3^T \\ & & B_3 & A_4 \end{bmatrix}$$

is the block tridiagonal, positive definite, symmetric stiffness matrix and X and Y (representing the generalized nodal displacements and loads, respectively) are partitioned correspondingly as

$$X = \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}, \qquad Y = \begin{Bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ Y_4 \end{Bmatrix}$$

This algorithm factors the total stiffness matrix into the product of a lower triangular array and its transpose and then solves a pair of triangular sets of equations. This factorization is possible only for positive definite matrices.

The method makes use of the Cholesky algorithm to factor the matrix A into the product of an upper and lower triangular matrix such that $A = LL^T$ with

$$L = \begin{bmatrix} L_1 & & & \\ M_1 & L_2 & & \\ & M_2 & L_3 & \\ & & M_3 & L_4 \end{bmatrix}$$

and

$$A_1 = L_1 L_1^T$$

$$B_1 = M_1 L_1^T \qquad \text{so that} \qquad M_1 = B_1 L_1^{-T}$$

$$A_2 = M_1 M_1^T + L_2 L_2^T \qquad \text{so that} \qquad L_2 L_2^T = A_2 - M_1 M_1^T$$

$$B_2 = M_2 L_2^T \qquad \text{so that} \qquad M_2 = B_2 L_2^{-T}$$

$$A_3 = M_2 M_2^T + L_3 L_3^T \qquad \text{so that} \qquad L_3 L_3^T = A_3 - M_2 M_2^T$$

etc.

These equations are used in turn to determine L_1, M_1, L_2, M_2, etc., obtaining each diagonal block by the Cholesky algorithm and each off-diagonal block by solving triangular equations.

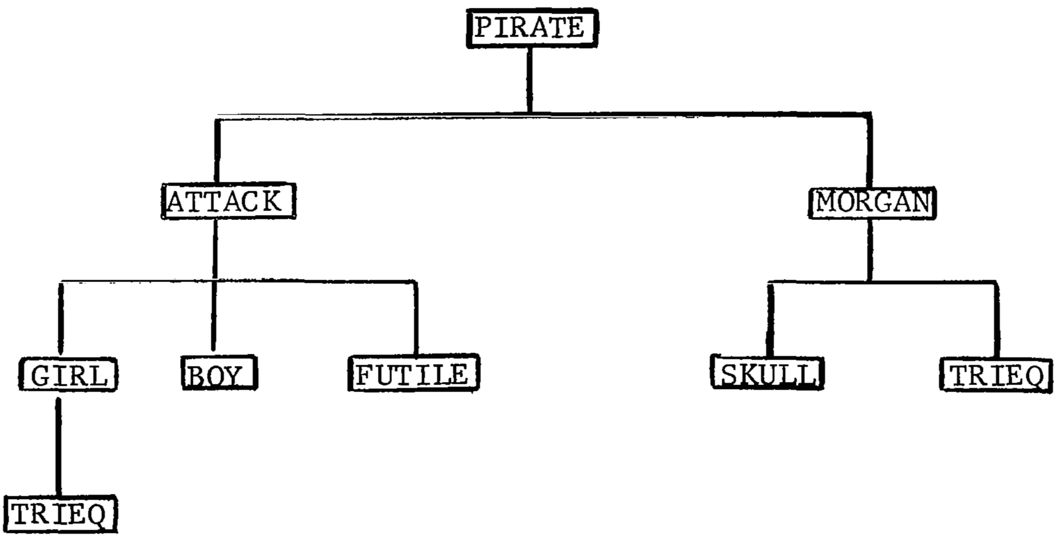
The diagonal blocks of the A matrix, A_i, are factored using the standard Cholesky formulas for each block

$$\ell_{kk} = \left[A_{kk} - \sum_{j=1}^{k-1} \ell_{kj}^2 \right]^{\frac{1}{2}}$$

and

$$\ell_{ik} = \left[A_{ik} - \sum_{j=1}^{k-1} \ell_{ij}\ell_{kj} \right] / \ell_{kk} \qquad \text{for} \qquad i > k$$

A flow chart for PIRATE is shown below.



The factorization is carried out by subroutine ATTACK as follows

Error

An error occurred while processing this page. See the system log for more details.

$$L_1Z_1 = Y_1$$
$$L_2Z_2 = Y_2 - M_1Z_1$$
$$L_3Z_3 = Y_3 - M_2Z_2$$
etc.

MORGAN calls SKULL to form the products M_kZ_k and TRIEQ to solve the triangular equations for the Z_k.

The "back" solution is so called because it obtains the solution elements in reverse order; in partitioned form the process is

$$L^TX = \begin{bmatrix} L_1^T & M_1^T & & \\ & L_2^T & M_2^T & \\ & & L_3^T & M_3^T \\ & & & L_4^T \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix} = \begin{Bmatrix} Z_1 \\ Z_2 \\ Z_3 \\ Z_4 \end{Bmatrix}$$

i.e.,

$$L_4^TX_4 = Z_4$$
$$L_3^TX_3 = Z_3 - M_3^TX_4$$
$$L_2^TX_2 = Z_2 - M_2^TX_3$$
etc.

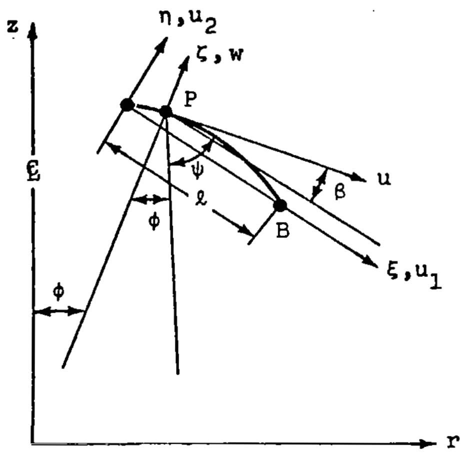
Once again, SKULL generates the products $M_k^TX_{k+1}$ and TRIEQ provides the solutions X_k. Since the X_k are obtained in reverse order, it is necessary for MORGAN to read them backwards in order to produce the solution in normal order in core. Note that this is done only after the initial solution since a "backwards" copy of the blocks of L are written on auxiliary storage for use in subsequent solutions that do not require the stiffness matrix to be factored.

4. ELEMENTS IN THE PLANS LIBRARY

In this section details about the theoretical aspects of the finite elements available in PLANS and their capabilities are presented. The order of discussion is based on the analytical and/or spatial dimensionality of the elements, i.e., one dimensional elements are discussed first, then two dimensional elements, and, finally, three dimensional elements. The topics discussed for each element are presented in the following sequence:

- Introduction
- Displacement Assumptions
- Formation of Stiffness Matrix
- Geometric Stiffness Matrix
- Initial Strain Stiffness Matrix (Plastic Load Vector)
- Stress Calculations
- Thermal Stress Calculations
- Material Properties
- Mechanical Loads
- General Comments

4.1 Axisymmetric Doubly Curved Shell Element



Displacement Assumptions:

$$\begin{Bmatrix} u \\ v \\ w \end{Bmatrix} = \begin{bmatrix} a_1 & a_2 & a_3 & a_4 \\ b_1 & b_2 & b_3 & b_4 \\ c_1 & c_2 & c_3 & c_4 \end{bmatrix} \begin{Bmatrix} 1 \\ \xi \\ \xi^2 \\ \xi^3 \end{Bmatrix}$$

Initial Strain Distribution:

$$\begin{Bmatrix} \varepsilon_s \\ \varepsilon_\theta \\ \gamma_{s\theta} \end{Bmatrix} = (1-\xi) \begin{Bmatrix} \varepsilon_s^{\,i}(\zeta) \\ \varepsilon_\theta^{\,i}(\zeta) \\ \gamma_{s\theta}^{\,i}(\zeta) \end{Bmatrix} + \xi \begin{Bmatrix} \varepsilon_s^{\,j}(\zeta) \\ \varepsilon_\theta^{\,j}(\zeta) \\ \gamma_{s\theta}^{\,j}(\zeta) \end{Bmatrix}$$

Fig. 5 Axisymmetric Thin-Shell Element

4.1.1 Introduction

This element is a doubly curved isoparametric thin shell element for the analysis of axisymmetrically loaded thin shells of revolution. It should be used for shells whose thickness to mean radius ratio is less than 1/10. The shell theory used to derive the element is Sanders' nonlinear theory for small strains and moderate rotations (Ref. 43). The element is based on the formulation initially presented by Levine and Armen (Ref. 44), and extended in Ref. 26. It has six degrees of freedom at each node and an axisymmetric torsion capability has been included. Thicknesses are specified at nodes and allowance is made for a linear thickness variation from node to node.

Two local coordinate systems are employed in the element derivation in addition to the global cylindrical r,θ,z system. One is a local rectangular Cartesian system ξ,θ,η, as shown in Fig. 5 with displacements u_1, u_2, v. The other is the shell curvilinear coordinate system defined by the middle surface displacement in the meridional direction, u, circumferential direction, v, and normal direction, w. Here ζ is the distance normal to the middle surface in the direction of w. The sign convention for stress and moment resultants and applied pressures is illustrated in Fig. 6.

4.1.2 Displacement Assumptions

The displacement field may be written in terms of the generalized Cartesian nodal displacements as follows

$$u_1 = H_{01}^{(1)}(\xi)u_{1_i} + H_{02}^{(1)}(\xi)u_{1_j} + H_{11}^{(1)}(\xi)u_{1,\xi_i} + H_{12}^{(1)}(\xi)u_{1,\xi_j}$$

$$u_2 = H_{01}^{(1)}(\xi)u_{2_i} + H_{02}^{(1)}(\xi)u_{1_j} + H_{11}^{(1)}(\xi)u_{2,\xi_i} + H_{12}^{(1)}(\xi)u_{2,\xi_j}$$

$$v = H_{01}^{(1)}(\xi)v_i + H_{02}^{(1)}(\xi)v_j + H_{11}^{(1)}(\xi)v_{,\xi_i} + H_{12}^{(1)}(\xi)v_{,\xi_j}$$

where $H_{01}^{(1)}(\xi)$, $H_{02}^{(1)}(\xi)$, $H_{11}^{(1)}(\xi)$, $H_{12}^{(1)}(\xi)$ are cubic Hermitian interpolation polynomials. This can be written in matrix form as

$$\{u_c\} = [N]\{u_{c_i}\} \tag{42}$$

This results in six degrees of freedom per node. These are the Cartesian displacements and their respective first derivatives. These can be related to the shell displacements, u,v,w, the rotation $\chi = dw/ds - u/R_1$, the linear meridional membrane

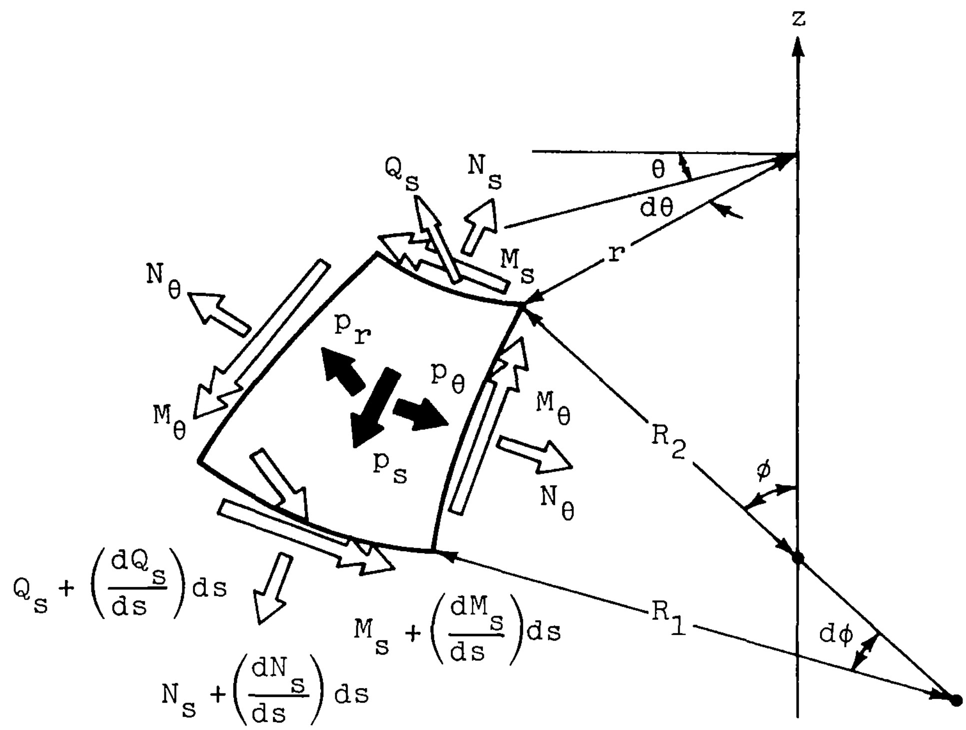


Fig. 6 Shell Forces and Moments - Axisymmetric Loading

strain $\epsilon_s^o = du/ds + w/R_1$ and the linear membrane shear strain $\gamma_{s\theta}^o = dv/ds - v \cos \varphi / r_o$, through the matrix [T]

$$\begin{Bmatrix} u_1 \\ u_2 \\ v \\ u_{1,\xi} \\ u_{2,\xi} \\ v_{,\xi} \end{Bmatrix} = \begin{bmatrix} \cos\beta & -\sin\beta & 0 & 0 & 0 & 0 \\ \sin\beta & \cos\beta & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & -\ell \tan\beta & \ell & 0 \\ 0 & 0 & 0 & \ell & \ell \tan\beta & 0 \\ 0 & 0 & \dfrac{\ell \cos\varphi}{r_o \cos\beta} & 0 & 0 & \dfrac{\ell}{\cos\beta} \end{bmatrix} \begin{Bmatrix} u \\ w \\ v \\ \chi \\ \epsilon_s^o \\ \gamma_{s\theta}^o \end{Bmatrix} \tag{43}$$

or

$$\{u_c\} = [T]\{u_s\}$$

The final nodal degrees of freedom $\{u_s\}$ are $u, w, \chi, \epsilon_s^o, v$, and $\gamma_{\varphi\theta}^o$ leading to a 12 degrees of freedom element. These quantities are assembled in the tangential-normal coordinate system at each node.

4.1.3 Formation of Stiffness Matrix

Based on Sanders' theory (Ref. 43) the linear components of the strain-displacement relations are

$$\begin{aligned} \epsilon_s &= \epsilon_s^o + \zeta\kappa_s \\ \epsilon_\theta &= \epsilon_\theta^o + \zeta\kappa_\theta \\ \gamma_{s\theta} &= \gamma_{s\theta}^o + 2\zeta\kappa_{s\theta} \end{aligned} \tag{44}$$

where

$$\epsilon_s^o = \frac{du}{ds} + \frac{w}{R_1}\ , \quad \epsilon_\theta^o = \frac{1}{r}\,(u \cos\varphi + w \sin\varphi)\ , \quad \gamma_{s\theta}^o = \frac{dv}{ds} - \frac{v \cos\varphi}{r_o}\ ,$$

$$\kappa_s = -\frac{d\chi}{ds}\ , \quad \kappa_\theta = -\frac{\cos\varphi}{r}\,\chi\ ,$$

$$\kappa_{s\theta} = \frac{1}{2r_o}\left\{\frac{1}{2}\frac{dv}{ds}\,(3 \sin\varphi - \frac{r_o}{R_1}) + \frac{v \cos\varphi}{2R_1}\,(1 - \frac{3R_1 \sin\varphi}{r_o})\right\}\ ,$$

$$\chi = \frac{dw}{ds} - \frac{u}{R_1}$$

The strain displacement relations may now be written in terms of the rectilinear displacements u_1, u_2, v and these are related to the nodal degrees of freedom to yield

$$\{\epsilon\} = \{\epsilon_o\} + \zeta\{\kappa\} = \{[W_m] + \zeta[W_b]\}\{u_s\} = [W]\{u_s\} \tag{45}$$

The elastic stiffness matrix may then be formed as

$$\begin{aligned}[k^0] &= \int_v [W]^T[E][W]dv \\ &= 2\pi\ell \int_o^1 \left\{h[W_m]^T[E][W_m] + \frac{h^3}{12}[W_b]^T[E][W_b]\right\} \frac{r_o d\xi}{\cos\beta}\end{aligned} \tag{46}$$

where $[E]$ is the matrix of elastic constants relating stress to strain. The quantity h is the shell thickness which varies linearly from node to node.

This line integral of Eq. (46) is evaluated numerically using Gauss-Legendre integration whose order is specified by the user. Up to eighth order integration is available but sixth order has been found to be sufficient for most purposes. For simple geometries and constant thickness, third order is adequate.

The actual geometry between nodes i and j is approximated by a substitute curve, represented in local ξ,η coordinates as $\eta = \eta(\xi)$. This curve matches the coordinates and slopes of the actual curve at the nodes of the element and is a cubic Hermitian polynomial

$$\eta = (2\xi^3 - 3\xi^2 + 1)\eta_1 + (3\xi^2 - 2\xi^3)\eta_2 + (\xi^3 - 2\xi^2 + \xi)\eta_1' + (\xi^3 - \xi^2)\eta_2' \tag{47}$$

The terms in parenthesis are the first order Hermitian polynomials $H_{01}^{(1)}(\xi)$, $H_{02}^{(1)}(\xi)$, $H_{11}^{(1)}(\xi)$, $H_{12}^{(1)}(\xi)$. Here η_1 and η_2

are the ordinates of the substitute curve at the end points and $\eta_1 = \eta_2 \equiv 0$. The terms η_1' and η_2' are the slopes $\tan\beta_i$ and $\tan\beta_j$. All appropriate geometric quantities such as η, R, and r_o are obtained from the R,z coordinates of points i and j and φ_i and φ_j, the angle between the outward normal and the positive z-axis.

4.1.4 Geometric Stiffness Matrix

The geometric nonlinear stiffness matrix is developed based on the small strain moderate rotation theory proposed by Sanders. It is intended for use with the "updated" or "convected" coordinate approach to the solution of geometric nonlinear problems. The approximate strain displacement equations specialized to axisymmetric problems are

$$\begin{aligned}
\epsilon_s &= \epsilon_s^o + \tfrac{1}{2}(\varphi_1^2 + \varphi^2) + \zeta k_s \\
\epsilon_\theta &= \epsilon_\theta^o + \tfrac{1}{2}(\varphi_2^2 + \varphi^2) + \zeta k_\theta \\
\gamma_{s\theta} &= \gamma_{s\theta}^o + \varphi_1\varphi_2 + 2\zeta k_{s\theta}
\end{aligned} \tag{48}$$

where

$$\varphi_1 = \frac{u}{R_1} - \frac{dw}{ds} = -\chi \quad , \quad \varphi_2 = \frac{v}{R_2}$$

$$\varphi = \frac{1}{2R_1 r_o}\,[(r_o v)_{,\varphi}]$$

We can write the geometric stiffness as

$$[k^1] = \int_A [\Omega]^T[N][\Omega]dA \tag{49}$$

where

$$\begin{Bmatrix} \varphi_1 \\ \varphi_2 \\ \varphi_3 \end{Bmatrix} = [\Omega]\{u_s\} \tag{50}$$

The stress resultants we assumed to vary linearly from node to node since stress computations are carried out at nodes

$$[N] = [N]_i(1-\xi) + [N]_j\xi \tag{51}$$

where

$$[N] = \begin{bmatrix} N_s & N_{s\theta} & 0 \\ N_{s\theta} & N_\theta & 0 \\ 0 & 0 & N_s+N_\theta \end{bmatrix}$$

This leads to

$$[k^1] = \int_A [\Omega]^T[N]_i[\Omega](1-\xi)dA + \int_A [\Omega]^T[N]_j[\Omega]\xi dA \tag{52}$$

where

$$dA = \frac{2\pi r_o \ell d\xi}{\cos\beta}$$

All area (line) integrations are performed using Gauss-Legendre integration of the same order as specified for the stiffness matrix.

4.1.5 Initial Strain Stiffness Matrix (Plastic Load Vector)

The effective plastic load vector can be written as

$$\{\Delta Q_i\} = \int_V [W]^T[E]\{\Delta\epsilon^P\}dv \tag{53}$$

The initial (plastic) strains are assumed to vary linearly from node to node while at the nodes the variation of the plastic strains through the thickness is arbitrary. We have then

$$\{\Delta\epsilon^P(\xi,\zeta)\} = \{\Delta\epsilon^P(0,\zeta)\}(1-\xi) + \{\Delta\epsilon^P(1,\zeta)\}\xi \tag{54}$$

Using this expression in Eq. (53) and remembering that $[W] = [W_m] + \zeta[W_b]$ the final expression for $\{\Delta Q_i\}$ is

$$\begin{aligned}
\{\Delta Q_i\} = & \int_A (1-\xi)[W_m]^T[E]dA \left[\int_{-h/2}^{h/2} \{\Delta\epsilon^P(0,\zeta)\}d\zeta\right] \\
& + \int_A \xi[W_m]^T[E]dA \left[\int_{-h/2}^{h/2} \{\Delta\epsilon^P(1,\zeta)\}d\zeta\right] \\
& + \int_A (1-\xi)[W_b]^T[E]dA \left[\int_{-h/2}^{h/2} \zeta\{\Delta\epsilon^P(0,\zeta)\}d\zeta\right] \\
& + \int_A \xi[W_b]^T[E]dA \left[\int_{-h/2}^{h/2} \zeta\{\Delta\epsilon^P(1,\zeta)\}d\zeta\right]
\end{aligned} \tag{55}$$

The integration of the effective plastic forces and moments through the thickness is carried out using Simpson's rule with an option of choosing up to 21 integration points (20 layers). The number of layers used must be an even number and is input by the user. Eleven points (10 layers) are usually sufficient. The area (line) integration is performed using a Gauss-Legendre integration scheme (input by the user) of the order used to form the stiffness matrix.

4.1.6 Stress Calculations

Stresses are calculated at the nodes of the element. All strain components are continuous from element to element except κ_s. This quantity is averaged at nodes. All stresses output are in the shell curvilinear coordinate system. Stresses are also calculated and output at each integration point through the thickness by node.

4.1.7 Thermal Stress Calculations

Orthotropic thermal stress calculations are allowed. Different thermal coefficients of expansion in the meridional and circumferential directions can be input. A parabolic temperature distribution through the thickness is assumed at nodes. Between nodes the meridional temperature variation is assumed to be linear. Temperatures are input at the top, bottom, and middle surface at each node. The thermal load vector is obtained from Eq. (55) where $\Delta\epsilon^P$ is replaced by $\alpha_i \Delta T$, i.e.,

$$\{\Delta\epsilon^P\} = \begin{Bmatrix} \alpha_s \Delta T \\ \alpha_\theta \Delta T \\ 0 \end{Bmatrix} \tag{56}$$

The integration of the thermal forces and moments is done exactly, assuming the parabolic temperature distribution.

Temperatures at layers are obtained by interpolating their values based on the parabolic distribution.

4.1.8 Material Properties

All material properties are constant within the element and temperature independent. Elastic properties are orthotropic. The meridional and circumferential directions are the principal directions of orthotropy. We have

$$\begin{Bmatrix} \sigma_s \\ \sigma_\theta \\ \tau_{s\theta} \end{Bmatrix} = \begin{bmatrix} \dfrac{E_s}{1-\nu_{s\theta}\nu_{\theta s}} & \dfrac{\nu_{s\theta}E_s}{1-\nu_{s\theta}\nu_{\theta s}} & 0 \\ \dfrac{\nu_{\theta s}E_\theta}{1-\nu_{s\theta}\nu_{\theta s}} & \dfrac{E_\theta}{1-\nu_{s\theta}\nu_{\theta s}} & 0 \\ 0 & 0 & G_{s\theta} \end{bmatrix} \begin{Bmatrix} \epsilon_s \\ \epsilon_\theta \\ \gamma_{s\theta} \end{Bmatrix} = [C]\{\epsilon\}$$

where $C_{11}C_{22} - C_{12}^2 > 0$ for positive definite stiffness matrices.

Plastic material properties are assumed to be orthotropic with Hill's yield criterion for plane stress used for initial yielding

$$f = (G+H)\sigma_s^2 + (F+H)\sigma_\theta^2 - 2H\sigma_s\sigma_\theta + 2N\tau_{s\theta}^2 = 1 \qquad (58)$$

with $G+H = 1/X^2$, $F+H = 1/Y^2$, $2H = 1/X^2 + 1/Y^2 - 1/Z^2$, $2N = 1/T^2$, and X,Y,Z are the yield stresses in tension in the s,θ,ζ directions, respectively. T is the yield stress in shear in the s-θ plane. For two dimensions the stability criterion for use of this yield condition reduces to

$$\frac{1}{X^2}\frac{1}{Y^2} - \frac{1}{4}\left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)^2 > 0 \qquad (59)$$

Perfectly plastic, linear strain hardening or nonlinear strain hardening laws may be chosen. For orthotropic plasticity the individual components of the stress-strain law are input and each component may be represented as linear or nonlinear strain hardening. If one component is perfectly plastic all must be. At the end of each half-cycle of loading new plastic material properties may be input but not new elastic material properties. This allows subsequent half-cycles of the stress-strain curve to be accurately represented.

Thermal coefficients of expansion may be input orthotropically, i.e., different in the meridional and circumferential directions. They may not be changed at the end of each half-cycle.

4.1.9 Mechanical Loads

Two types of mechanical loads may be applied to a shell element. They are surface tractions and concentrated (or line) loads.

Concentrated or Line Loads (Edge Loads) — These are applied as forces per unit length of circumference except at $r = 0$ where $N_s = M_s = N_{s\theta} = 0$ and $F_z \equiv Q_s$ represents an actual force. At any other radial location the line (or edge) loads are applied in the natural (shell) curvilinear coordinates as N_s, Q_s, M_s, and $N_{s\theta}$ in the u, w, χ, and v directions, respectively. These forces per unit length are applied and specified at nodes of the shell.

Surface Loads — The surface tractions are specified per unit surface area in the meridional p_s, normal, p_ζ, and circumferential directions, p_θ. They are assumed to vary linearly from node to node. All three components at each node must be specified. The loads are applied through consistent load vectors which are calculated in the program

$$\{P_i\} = 2\pi\ell\left[\int_0^1 [N]^T r d\xi \begin{Bmatrix} p_{s_i} \\ p_{\zeta_i} \\ p_{\theta_i} \end{Bmatrix} + \int_0^1 [N]^T \xi r d\xi \begin{Bmatrix} p_{s_j} - p_{s_i} \\ p_{\zeta_j} - p_{\zeta_i} \\ p_{\theta_j} - p_{\theta_i} \end{Bmatrix}\right] \tag{60}$$

All integrations are carried out numerically using a Gauss-Legendre scheme of the same order as specified for the stiffness matrix.

4.1.10 Equilibrium Correction

The equilibrium correction term is written as

$$\{R_i\} = \{P_i\} - \int_V [W]^T\{\sigma\}dv = \{P_i\} - \left[\int_A [W]^T(1-\xi)\{N_i\}dA + \int_A [W]^T\xi\{N_j\}dA\right] \tag{61}$$

Here the stresses are assumed to vary linearly from node to node for the equilibrium correction. The stress and moment resultant vectors at each node $\{N_i\}$ are evaluated by numerically integrating the stresses through the thickness using Simpson's rule in the same manner used to obtain the "inelastic forces and moments."

4.1.11 General Comments

- Pole Conditions – At a pole $(r = 0)$ the following conditions must be satisfied for the shell element: 1) $u_R = 0$, if we have a shell where $dr/dz = 0$ at $r = 0$ (i.e., $\varphi = 0$ or π) then this implies $u = 0$; 2) $\chi = 0$; 3) $v = 0$; and 4) $\gamma_{\varphi\theta} = 0$. Correspondingly, no forces can be applied at $r = 0$, i.e., $N_s = M_s = N_{s\theta} \equiv 0$.

- Cap Element – No special cap element was required for the axisymmetric shell element described here. The use of the Hermitian polynomial representation of the displacement components u_1 and u_2 eliminates the necessity of obtaining a relationship between the generalized coordinates and the degrees of freedom anywhere in the element, and specifically at a pole. Furthermore, since the integration was performed numerically using the Gauss-Legendre technique, no actual evaluation of the function matrix [W] at the point of singularity or indeterminacy was required. The singularity was isolated and all improper integrals required for the various matrices converge. The terms of the matrix [W] required for the u_i and χ_i degrees of freedom, which might result in numerical problems, were removed since the radial displacement and rotation are identically zero at the pole. To obtain strains at the pole, appropriate limits were taken in the [W] matrices, employing L'Hopital's rule. It is, however, necessary to use a finer grid in the region of the pole than elsewhere. Treating the pole as if it were an actual physical boundary leads to accurate results for displacements and stresses.

- Geometric nonlinear analysis using this shell element is only available in a special purpose module called AXSHEL. It is not available in REVBY at the release date of this report.

- This element is available in the REVBY module of PLANS.

4.2 Axisymmetric Ring Element

4.2.1 Introduction

Classical beam theory forms the basis for the development of the stiffness matrix for a thin circular ring of arbitrary cross section, as derived in this section. This element is used in the REVBY program and is intended for use with the revolved axisymmetric shell element. Pertinent geometry defining the ring-shell intersection is shown in Fig. 7. In the following derivations it is assumed that the shear center and centroid of the ring coincide.

4.2.2 Displacement Assumptions and Formation of Stiffness Matrix

Based on thin ring theory for an axisymmetric circular ring, the total strain and stress for any point in the cross section in the presence of an initial (plastic) strain is

$$e = -\frac{1}{r_c}(u_c - y\beta_z) \tag{62}$$

$$\sigma = E(e - \epsilon) \tag{63}$$

where r_c is the radius of the ring, ϵ is an initial (plastic) strain, u_c is a component of displacement of the ring axis, and β_z is a rotation normal to the ring cross section.

Substituting Eqs. (62) and (63) into the expression for strain energy leads to

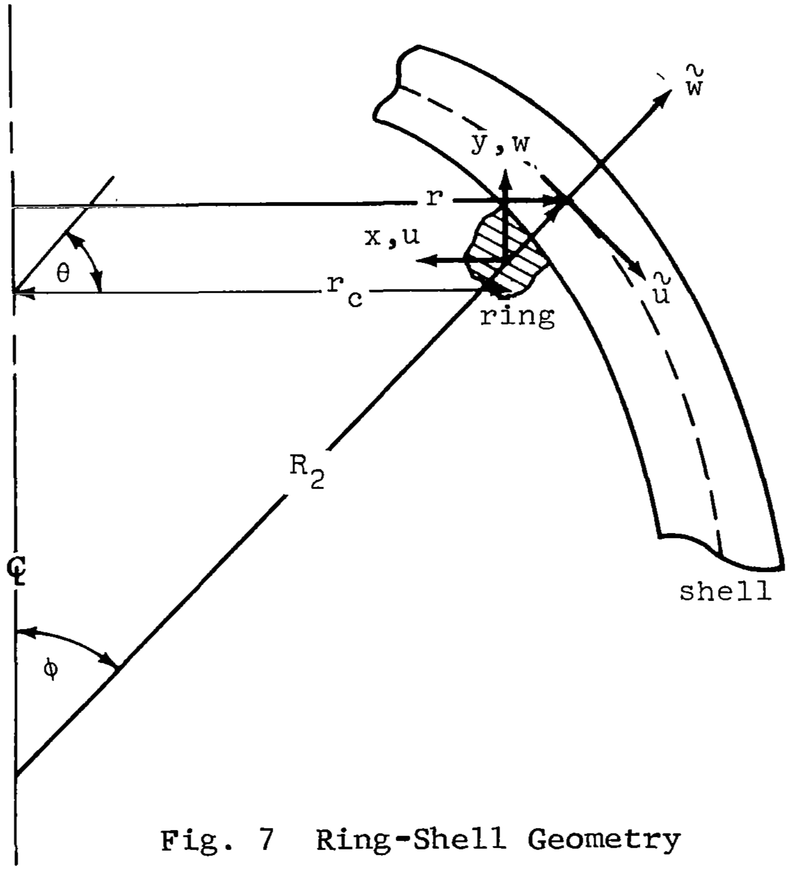


Fig. 7 Ring-Shell Geometry

$$U = \frac{E\pi}{r_c}\,(Au_c^2 + I_x\beta_z^2) + 2\pi E \int_v (u_c - y\beta_z)\epsilon dA + \frac{E}{2}\int_v \epsilon^2 dv \qquad (64)$$

where A and I_x are the section area and moment of inertia. Using the Principal of Virtual Work yields the equilibrium equation relating force, displacement, and initial strains

$$f = [k]\begin{Bmatrix} u_c \\ \beta_z \end{Bmatrix} - [k^*]\begin{Bmatrix} \int_A \epsilon dA \\ \int_A y\epsilon dA \end{Bmatrix} \qquad (65)$$

where $[k]$ is the element stiffness matrix given by

$$[k] = \frac{2}{r_c} \left[\begin{array}{c|c} EA & 0 \\ \hline 0 & EI_x \end{array}\right] \tag{66}$$

and $[k^*]$ is the initial strain stiffness matrix.

4.2.3 Initial Strain Stiffness Matrix

The initial strain stiffness matrix obtained in Eq. (65) is given by

$$[k^*] = 2\pi E \left[\begin{array}{c|c} -1 & 0 \\ \hline 0 & 1 \end{array}\right] \tag{67}$$

The plastic load vector $\{Q\}$ is the product of $[k^*]$ and the vector of integrated plastic strains.

Since the plastic strain may vary through the ring cross section the integrals in Eq. (64) cannot be determined a priori. These integrals are performed numerically for plastic rings using a Gaussian quadrature.

4.2.4 Transformation to Shell Coordinate System

The stiffness matrices of Eqs. (66) and (67) are written with respect to the ring coordinate system. In order to use the ring element with the axisymmetric shell element the ring must be related to the shell coordinate system.

The pertinent geometry defining the ring-shell intersection is shown in Fig. 8 for an attachment at point A. Continuity of the ring and shell displacements at point A leads to

$$\begin{Bmatrix} u_c \\ \beta_z \end{Bmatrix} = [T] \begin{Bmatrix} \tilde{u} \\ \tilde{w} \\ \tilde{\phi}_1 \end{Bmatrix} \tag{68}$$

where

$$[T] = \left[\begin{array}{c|c|c} -r' & -\dfrac{r}{R_2} & e_y \dfrac{r}{R_2} - z_o r' \\ \hline 0 & 0 & -1 \end{array}\right]$$

$\tilde{u}$, $\tilde{w}$, $\tilde{\phi}_1$ are the shell displacements and rotation, r and R_2 are the shell radii of curvature shown in Fig. 8, and "prime" denotes differentiation with respect to the arc length, s.

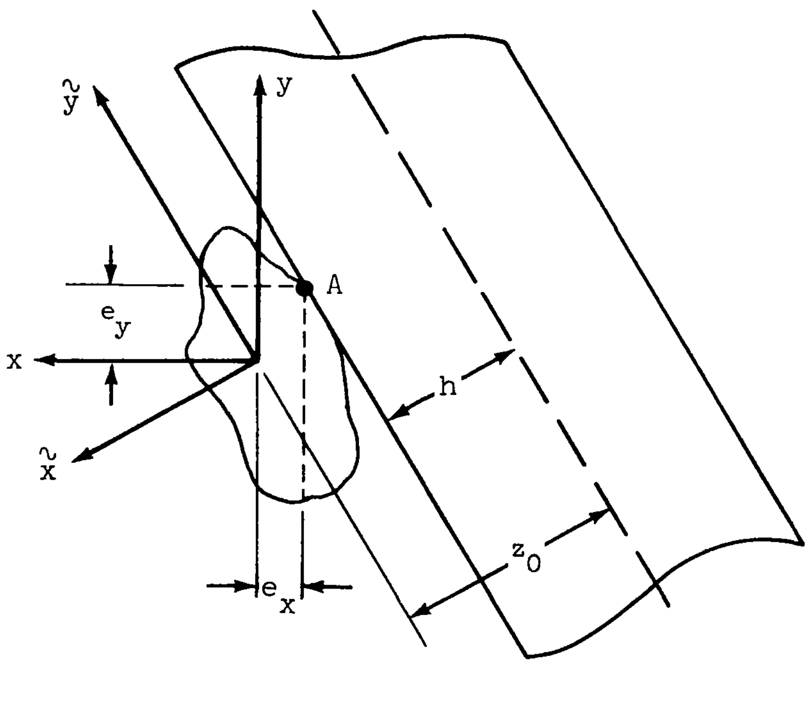


Fig. 8 Geometry Defining the Ring-Shell Intersection

The final stiffness and initial strain matrix with respect to the shell displacements then is

$$[k_s] = [T]^T[k][T] \tag{69}$$

$$[k_s^*] = [T]^T[k^*] \tag{70}$$

4.2.5 Stress Calculations

Stresses are monitored at a specified number of Gauss points for each cross section via Eq. (63). From these stress values, each point is checked for yielding and if necessary plastic strains are calculated. Once these values are determined the numerical integration can be performed.

Five distinct ring cross sections are currently in the REVBY module, a solid rectangle, solid circular, Z-section, I-section, and hollow circular section (see general comments for section details).

4.2.6 Thermal Stress Analysis

A parabolic temperature distribution can be specified through the depth of the ring (i.e., along $\tilde{x}$ in Fig. 8). Three temperatures are specified. They are at the top, centroid, and bottom of the ring. The thermal load vector is formed in the same manner as the plastic load vector with ϵ replaced by $\alpha\Delta T$. Thermal strains at integration points are determined by interpolation based on the assumed parabolic distribution.

4.2.7 Material Properties

Elastic, ideally plastic, linear strain hardening or nonlinear strain hardening stress-strain laws may be chosen. The nonlinear hardening is based on a Ramberg-Osgood representation of the actual stress-strain data. The plastic material properties may be changed at the end of each half-cycle of loading.

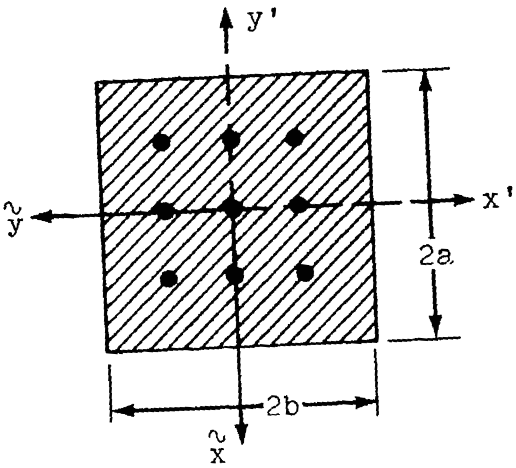
4.2.8 Loads

Only axisymmetric line loads may be applied at the node defining the ring location.

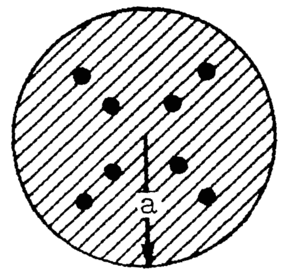
4.2.9 General Comments

- Five different cross sectional shapes for the ring are available in the REVBY module. These are shown in Fig. 9. They are: 1) a solid rectangular cross section, SREC; 2) a solid circular cross section, SCIR; 3) a Z-section with equal area flanges, i.e., $a = b$, $t_1 = t_2$, ZSEC; 4) an I-section with equal area flanges, i.e., $a = b$, $t_1 = t_2$, ISEC; and 5) a hollow circular cross section, HCIR. The order of the Gauss-Legendre integration schemes used to obtain the plastic load vector follows

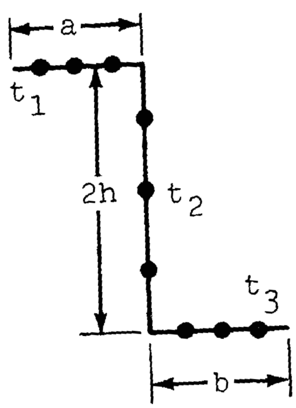
Section	Gauss-Legendre Integration Order
SREC	Three by three array (9 points)
SCIR	Two radial points times four circumferential points (8 points)
ZSEC	Third order in each flange and web (9 points)
ISEC	Third order in each flange and web (9 points)
HCIR	Eighth order around the circumference (8 points)



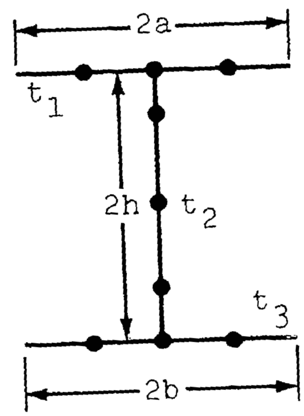
SREC
Solid Rectangular
Section



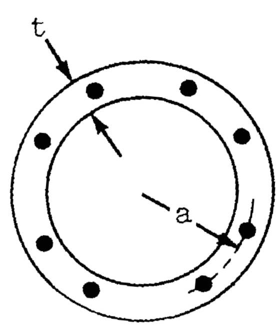
SCIR
Solid Circular
Section



ZSEC
Z-Section



ISEC
I-Section



HCIR
Hollow Circular
Section

Fig. 9 Available Ring Cross Sections in REVBY

- For the Z- and I-sections the thicknesses are assumed to be small compared to the flange and web lengths. For the hollow circular section it is assumed $t/a \ll 1$.
- The applicable module for this element is REVBY.

4.3 Stringer Element

4.3.1 Introduction

This element is used to represent a one dimensional axial force structural member. Two stringer elements are included in the PLANS system: a two-node element developed from a constant axial strain assumption and a three-node element developed from a linearly varying strain assumption (Fig. 10).

4.3.2 Displacement Assumption

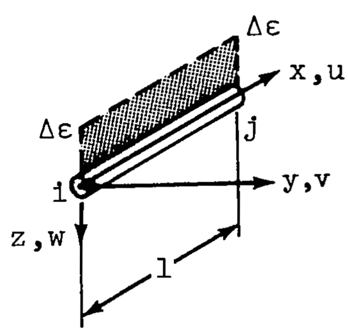
The distribution of the three local displacements u,v,w shown in Fig. 10 is assumed to be

$$\begin{Bmatrix} u(x) \\ v(x) \\ w(x) \end{Bmatrix} = (1-\xi)\begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} + \xi\begin{Bmatrix} u_j \\ v_j \\ w_j \end{Bmatrix} \tag{71}$$

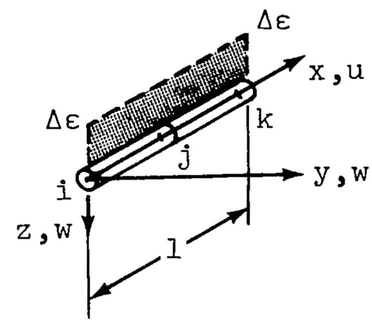
where $\xi = x/\ell$ for the two-node stringer and

$$\begin{Bmatrix} u(x) \\ v(x) \\ w(x) \end{Bmatrix} = (2\xi^2 - 3\xi + 1)\begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} + (2\xi^2 - \xi)\begin{Bmatrix} u_j \\ v_j \\ w_j \end{Bmatrix} + 4(\xi - \xi^2)\begin{Bmatrix} u_k \\ v_k \\ w_k \end{Bmatrix} \tag{72}$$

for the three-node stringer.



(a) Constant strain element



(b) Linear strain element

Displacement Assumption:

$u = a_1 + a_2 x$ $\qquad\qquad$ $u = a_1 + a_2 x + a_3 x^2$

Initial Strain Distribution

ε = constant

Fig. 10 Stringer Element

As shown in Fig. 10, the indices i,j denote the end point nodes of the three-node stringer. The specification of the axial displacement component is sufficient to fully describe the elastic linear response of a stringer element since the element has no out-of-plane (normal to the axial direction) stiffness. The normal components, $v,w,$ contribute to the elastic, geometric nonlinear response by coupling the axial force to the out-of-plane response. This coupling leads to the initial stress stiffness matrix $[k^1]$. It should be noted that displacement components $v,w,$ as shown in Fig. 10, refer to any two mutually perpendicular directions normal to the stringer axis since the stringer does not have a preferential cross sectional reference system.

4.3.3 Element Stiffness Matrix

The element stiffness matrices are obtained from the principle of virtual work (Section 3). The appropriate strain displacement relations are

$$\epsilon_x = \frac{\partial u}{\partial x} + \frac{1}{2}\left(\frac{\partial w}{\partial x}\right)^2 + \frac{1}{2}\left(\frac{\partial v}{\partial x}\right)^2 \tag{73}$$

Using this approach, the linear and nonlinear components of total strain are obtained from Eqs. (71) and (73) as

$$\Delta e = \frac{1}{\ell} \left[-1 \;\vdots\; 1 \right] \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} \tag{74}$$

$$\begin{Bmatrix} v_{,x} \\ w_{,x} \end{Bmatrix} = \frac{1}{\ell} \left[\begin{array}{c|c|c|c} -1 & +1 & 0 & 0 \\ \hline 0 & 0 & -1 & +1 \end{array} \right] \begin{Bmatrix} v_i \\ v_j \\ w_i \\ w_j \end{Bmatrix} \tag{75}$$

for the two-node stringer and

$$\Delta e = \frac{1}{\ell} \left[4\xi-3 \;\vdots\; 4\xi-1 \;\vdots\; 8\xi-4 \right] \begin{Bmatrix} u_i \\ u_j \\ u_k \end{Bmatrix} \tag{76}$$

$$\begin{Bmatrix} v_{,x} \\ w_{,x} \end{Bmatrix} = \frac{1}{\ell}\left[\begin{array}{c:c:c:c:c:c} 4\xi-3 & 4\xi-1 & 8\xi-4 & 0 & 0 & 0 \\ \hdashline 0 & 0 & 0 & 4\xi-3 & 4\xi-1 & 8\xi-4 \end{array}\right]\begin{Bmatrix} v_i \\ v_j \\ v_k \\ w_i \\ w_j \\ w_k \end{Bmatrix} \tag{77}$$

for the three-node stringer.

Making use of Eqs. (74) and (76) in the definitions for the elastic stiffness matrix $[k^0]$ and assuming a constant cross section stringer then yields

$$[k^0] = \frac{AE}{\ell}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad \text{for the two-node stringer} \tag{78}$$

and

$$[k^0] = \frac{AE}{3\ell}\begin{bmatrix} 7 & -8 & 1 \\ -8 & 16 & -8 \\ 1 & -8 & 7 \end{bmatrix} \quad \text{for the three-node stringer} \tag{79}$$

where A is the section area and E is Young's modulus.

4.3.4 Geometric Stiffness Matrix

Using Eqs. (75) and (77) in the definition of the geometric stiffness $[k^1]$ gives

$$[k^1] = \frac{\bar{F}}{\ell}\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 1 & 0 & 0 \\ 0 & 0 & 1 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \quad \text{for the two-node stringer} \tag{80}$$

and

$$[k^1] = \frac{\bar{F}}{3\ell}\begin{bmatrix} 7 & 1 & -8 & 0 & 0 & 0 \\ 1 & 7 & -8 & 0 & 0 & 0 \\ -8 & -8 & 16 & 0 & 0 & 0 \\ 0 & 0 & 0 & 7 & 1 & -8 \\ 0 & 0 & 0 & 1 & 7 & -8 \\ 0 & 0 & 0 & -8 & -8 & 16 \end{bmatrix} \quad \text{for the three-node stringer} \tag{81}$$

Here $\bar{F}$ is the average axial force.

4.3.5 Initial Strain Stiffness Matrix

The initial strain matrix is based on the assumption that the thermal (plastic) strain is constant in the element. Thus, the initial strain matrix can be written

$$[k^*] = EA \begin{bmatrix} -1 \\ +1 \\ 0 \end{bmatrix} \tag{82}$$

for both the two- and three-node stringer elements.

4.3.6 Stress and Force Calculations

Element axial forces in the presence of thermal (plastic) strains are calculated as

$$\{f\} = [k^0]\{\delta\} - [k^*]\{\epsilon_p\} \tag{83}$$

where $\{\delta\}$ are the local axial displacements and $\{\epsilon_p\}$ are the average thermal and/or plastic strains. Average element stresses are calculated at the center of the element and are used in conjunction with plasticity calculations consistent with the assumption of constant plastic strain within each element.

4.3.7 Thermal Stress Calculations

Temperatures are specified at the two end point nodes. These nodal values are used to determine an average element temperature.

4.3.8 Material Properties

Plastic strains are calculated using a uniaxial bilinear stress-strain relation or a nonlinear Ramberg-Osgood relation. Perfect plasticity is also accommodated.

4.3.9 Loads

The stringer can be loaded by concentrated nodal loads or consistent distributed line loads (Section 4.7.9).

4.3.10 Comments

- The three-node stringer has an assumed displacement function consistent with that used for the LST of Section 4.7 and is, therefore, used when linking an LST with a stringer.
- The applicable modules for this element are OUT-OF-PLANE, BEND, and OUT-OF-PLANE-MG.

4.4 Beam Element

4.4.1 Introduction

The stiffness properties for an initially straight beam finite element of arbitrary cross section are derived in this section based on the assumptions of classical beam theory. These assumptions include

1) Normals to the centroidal axis remain straight and normal after deformation and their length remains unchanged, i.e., the effect of transverse shear deformation and transverse normal strains are neglected.

2) Warping of the cross section is nelgected.

Although assumption 2) is restrictive for all but circular cross sections, the ability to specify the actual torsional rigidity has been maintained. Throughout the development, it is assumed that the shear center and centroid of the cross section do not necessarily coincide. The element coordinate system (located at the shear center of the cross section) and convention for moments, forces, and displacements are shown in Fig. 11. Based on

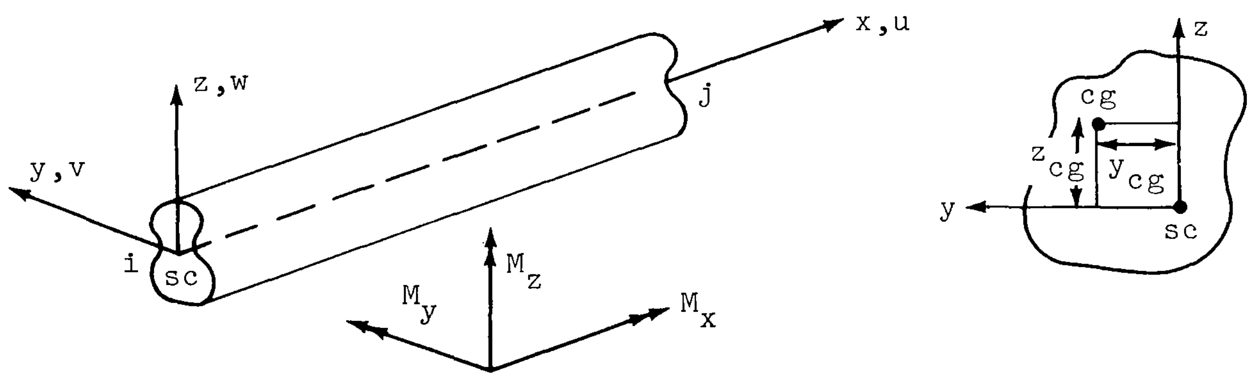


Fig. 11 Coordinate System and Convention for Forces and Displacements for Three Dimensional Beam Element

the convention of Fig. 11 and assumptions 1) and 2), as noted previously, the displacements at any point within the beam cross section can be written as

$$
\begin{aligned}
u_z &= u_c + (z - z_{cg})\beta_y - (y - y_{cg})\beta_z \\
v_z &= v_s - z\beta_x \\
w_z &= w_s + y\beta_x
\end{aligned}
\tag{84}
$$

where u_c is the axial displacement of the centroidal axis; v_s, w_s are the lateral displacements of the shear center in the cross sectional y and z directions, β_x, β_y, β_z are cross sectional rotations defined in Fig. 12, and y_{cg}, z_{cg} are the distances between the centroid and shear center.

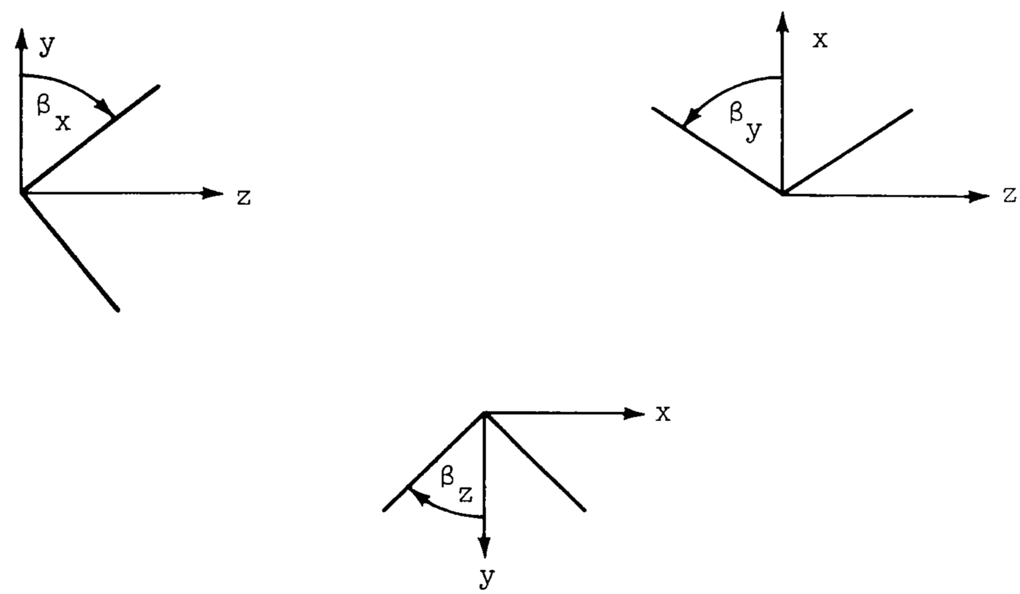


Fig. 12 Convention for Rotations

Based on Eqs. (84) and excluding terms associated with nonlinear terms in the curvatures, the nonlinear strain displacement relations are

$$
\begin{aligned}
e_x &= e_o + (z - z_{cg})\kappa_y - (y - y_{cg})\kappa_z - \epsilon_x^p \\
\gamma_{xy} &= - \beta_x\beta_y - z\kappa_{yz} - \gamma_{xy}^p \\
\gamma_{xz} &= y\kappa_{yz} - \beta_x\beta_z - \gamma_{xz}^p
\end{aligned}
\tag{85}
$$

where

$$e_o = \frac{\partial u_c}{\partial x} + \tfrac{1}{2}\beta_y^2 + \tfrac{1}{2}\beta_z^2$$

$$\kappa_y = \frac{\partial^2 w}{\partial x^2} \quad ; \quad \kappa_z = \frac{\partial^2 v}{\partial x^2}$$

$$\kappa_{yz} = \frac{\partial \beta_x}{\partial x}$$

and ϵ_x^p, γ_{xz}^p, γ_{xy}^p are initial or plastic strains.

Equations (85) along with the definitions for moment and product of inertia about the section centroid

$$I_y = \int_A (z - z_{cg})^2 dA \quad , \quad I_z = \int_A (y - y_{cg})^2 dA$$

$$I_{yz} = \int_A (y - y_{cg})(z - z_{cg}) dA$$

are used to derive the element stiffness matrices.

4.4.2 Displacement Assumptions

There are four independent displacement components that are prescribed to formulate the beam element stiffness properties. These are the two transverse displacement components, v_s, w_s and the axial displacement and twist, u_c and β_x. The out-of-plane rotations are related to the transverse displacements in the usual manner by imposing Kirchoff's hypothesis, i.e.,

$$\beta_y = -w_{,x} \quad , \quad \beta_z = v_{,x}$$

the two components of transverse displacement are assumed to be cubic functions that are solutions to the linear homogeneous beam equations

$$\begin{Bmatrix} v_s \\ w_s \end{Bmatrix} = \sum_{i=1}^{2} H_{0i}^{(1)} \begin{Bmatrix} v_i \\ w_i \end{Bmatrix} + H_{1i}^{(1)} \ell \begin{Bmatrix} v_{,x_i} \\ w_{,x_i} \end{Bmatrix} \tag{86}$$

where $\xi = x/\ell$, ℓ is the element length, and

$$H_{01}^{(1)} = 2\xi^3 - 3\xi^2 + 1$$

$$H_{02}^{(1)} = -2\xi^3 + 3\xi^2$$

$$H_{11}^{(1)} = \xi^3 - 2\xi^2 + \xi$$

$$H_{12}^{(1)} = \xi^3 - \xi^2$$

It is assumed that the in-plane displacement and twist u_c, β_x are linear functions of the axial coordinate

$$\begin{Bmatrix} u_c \\ \beta_x \end{Bmatrix} = \sum_{i=1}^{2} H_{0i}^{(0)} \begin{Bmatrix} u_i \\ \beta_{x_i} \end{Bmatrix} \tag{87}$$

where

$$H_{01}^{(0)} = 1 - \xi$$

$$H_{02}^{(0)} = \xi$$

4.4.3 Stiffness Matrix

The elastic component of the stiffness matrix can be developed using the principle of virtual work or from energy considerations. In either case, use is made of the kinematic displacement assumptions, Eqs. (84), to reduce the general relationship for the stiffness matrix, which is a function of the three spatial coordinates and involves a volume integral, to a function of only the axial coordinate and a simple one dimensional integral of the form

$$[k^0] = \int_0^{\ell} [W]^T[D][W]dx$$

where

$$[D] = \begin{bmatrix} EA & 0 & 0 & 0 \\ 0 & GJ & 0 & 0 \\ 0 & 0 & EI_{yy} & -EI_{yz} \\ 0 & 0 & -EI_{yz} & EI_{zz} \end{bmatrix}$$

and A is the cross sectional area, I_{yy}, I_{zz}, I_{yz} the centroidal moments and product of inertia, respectively, J is the section torsional rigidity, and E and G are Young's modulus and the shear modulus, respectively. The [W] matrix is obtained by substituting the displacement assumptions, Eqs. (86) and (87) into the linear component of the strain displacement relations, Eq. (85) and is written as

$$[W] = \begin{bmatrix} -\frac{1}{\ell} & 0 & 0 & 0 & 0 & 0 & \frac{1}{\ell} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -\frac{1}{\ell} & 0 & 0 & 0 & 0 & 0 & \frac{1}{\ell} & 0 & 0 \\ 0 & 0 & -\frac{\phi_1}{\ell^2} & 0 & -\frac{\phi_2}{\ell} & 0 & 0 & 0 & \frac{\phi_1}{\ell^2} & 0 & \frac{\phi_3}{\ell} & 0 \\ 0 & \frac{\phi_1}{\ell^2} & 0 & 0 & 0 & \frac{\phi_2}{\ell} & 0 & -\frac{\phi_1}{\ell^2} & 0 & 0 & 0 & \frac{\phi_3}{\ell} \end{bmatrix} \tag{88}$$

where

$$\phi_1 = 6(2\xi - 1) \quad , \quad \phi_2 = (6\xi - 4)$$

$$\phi_3 = (6\xi - 2)$$

The resulting stiffness matrix is a 12 x 12 matrix with six degrees of freedom per node u_i, v_i, w_i, β_{x_i}, β_{y_i}, β_{z_i}, where the axial displacement refers to the centroid and the lateral displacements refer to the shear center.

4.4.4 Geometric Stiffness Matrix

The development of the "geometric" stiffness matrix or initial stress stiffness matrix, based on small strain, moderate rotation assumptions for a planar beam, is straightforward and has been previously presented in Ref. 17. However, for the case of a general space frame subjected to torsion as well as bending, there is nonlinear coupling between bending and torsion which can lead to various expressions for the initial stress stiffness matrix depending on the approximations made in arriving at the strain displacement relations, Eq. (85). In this development the simplest approach was taken in that nonlinear contributions to the curvature and twist are neglected and Kirchoff's hypothesis is imposed on only the linear components of strain. In so doing the form of $[k^1]$ is similar to that for a planar beam with the addition of only the nonlinear coupling between torsion and bending.

With these considerations, $[k^1]$ can be written as

$$[k^1] = \int_0^\ell [\Omega]^T [\Sigma][\Omega] dx \tag{89}$$

where

$$[\Sigma] = \begin{bmatrix} 0 & -V_y & -V_z \\ -V_y & T & 0 \\ -V_z & 0 & T \end{bmatrix}$$

and

$$[\Omega] = \begin{bmatrix} 0 & 0 & 0 & \phi_4 & 0 & 0 & 0 & 0 & 0 & \phi_5 & 0 & 0 \\ 0 & 0 & -\frac{\phi_1}{\ell} & 0 & \phi_2 & 0 & 0 & 0 & \phi_1 & 0 & \phi_3 & 0 \\ 0 & \frac{\phi_1}{\ell} & 0 & 0 & 0 & \phi_2 & 0 & -\frac{\phi_1}{\ell} & 0 & 0 & 0 & \phi_3 \end{bmatrix}$$

$$\phi_1 = 6(\xi^2 - \xi) \quad , \quad \phi_2 = 3\xi^2 - 4\xi + 1 \quad , \quad \phi_3 = 3\xi^2 - 2\xi$$

$$\phi_4 = 1 - \xi \quad , \quad \phi_5 = \xi$$

4.4.5 Initial Strain Stiffness Matrix

The initial strain stiffness matrix is used to compute the nodal forces due to thermal or initial strains. In a plastic analysis using the "initial strain" method it is used to determine the effective plastic load vector (Ref. 8) as

$$\{\Delta Q\} = [k^*]\{\Delta\epsilon\}$$

where

$\{\Delta Q\}$ is the effective plastic load vector

$[k^*]$ is the initial strain stiffness matrix

and

$\{\Delta\epsilon\}$ is the vector of discrete quantities characterizing the state of plastic strain.

The initial strain stiffness matrix, based on the displacement assumptions of Eq. (84) can be written as

$$[k^*] = - \int_V [W]^T[\bar{E}] \begin{Bmatrix} \Delta\epsilon_x \\ (z - z_{cg})\Delta\epsilon_x \\ (y - y_{cg})\Delta\epsilon_x \\ y\Delta\gamma^p_{xz} - z\Delta\gamma^p_{yz} \end{Bmatrix} dV \qquad (90)$$

where $[W]$ is the matrix defined in Eq. (88) and

$$[\bar{E}] = \begin{bmatrix} E & 0 & 0 & 0 \\ 0 & E & 0 & 0 \\ 0 & 0 & -E & 0 \\ 0 & 0 & 0 & G \end{bmatrix}$$

Because for a three dimensional space frame, the plastic strain varies in each cross section as well as in the axial direction, Eq. (90) cannot be immediately reduced to a one dimensional integral as was the case for the stiffness matrix. However, consistent with finite element analysis, the functional form for the distribution of plastic strain increment can be assumed for each beam element. To this end, it is assumed that each component of increment of plastic strain varies linearly in the beam axial direction. Thus

$$\begin{aligned} \epsilon_x &= (1 - \tfrac{x}{\ell})\epsilon_{x_i}(y,z) + \tfrac{x}{\ell}\,\epsilon_{x_j}(y,z) \\ \gamma^p_{xy} &= (1 - \tfrac{x}{\ell})\gamma^p_{xy_i}(y,z) + \tfrac{x}{\ell}\,\gamma^p_{xy_j}(y,z) \\ \gamma^p_{xz} &= (1 - \tfrac{x}{\ell})\gamma^p_{xz_i}(y,z) + \tfrac{x}{\ell}\,\gamma^p_{xz_j}(y,z) \end{aligned} \tag{91}$$

where the i,j denote the two beam nodes, respectively (Fig. 11). At this point no assumption is made for the distribution in any cross section. With this assumption, $[k^*]$ can be written as

$$[k^*] = -\int_0^{\ell} [W]^T[\bar{E}][\bar{C}_1 \;\vdots\; \bar{C}_2]dx \int_A \left\{ \begin{array}{c} \Delta\epsilon_{x_i} \\ \cdot \\ \cdot \\ \cdot \\ \hline \Delta\epsilon_{x_j} \\ \cdot \\ \cdot \\ \cdot \\ y\Delta\gamma^p_{xz_j} - z\Delta\gamma^p_{xy_j} \end{array} \right\} dA \qquad (92)$$

where

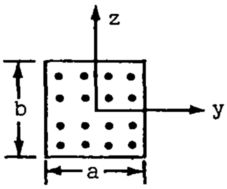
$$[\bar{C}_i] = \begin{bmatrix} \phi_i^{(0)} & 0 & 0 & 0 \\ 0 & \phi_i^{(0)} & 0 & 0 \\ 0 & 0 & \phi_i^{(0)} & 0 \\ 0 & 0 & 0 & \phi_i^{(0)} \end{bmatrix}$$

$i = 1, 2$

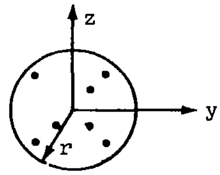
and

$$\phi_1^{(0)} = (1 - \frac{x}{\ell}) \quad ; \quad \phi_2^{(0)} = \frac{x}{\ell}$$

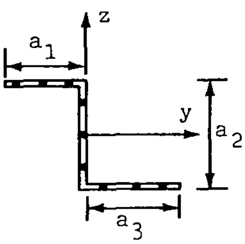
The integrals in Eq. (92) are evaluated numerically using a Gauss-Legendre integration scheme. To accomplish this, the shape of the cross section must be known a priori and increments of plastic strain must be evaluated at the Gauss points in the cross section. To this end nine distinct cross sections have been provided for in PLANS. These are shown in Fig. 13 along with the order of Gauss-Legendre integration.



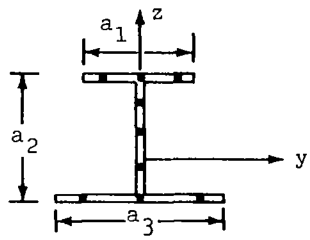
Solid Rectangular Section
16 Integration Points



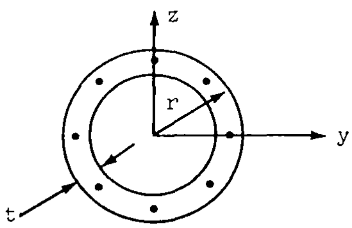
Solid Circular Section
8 Integration Points



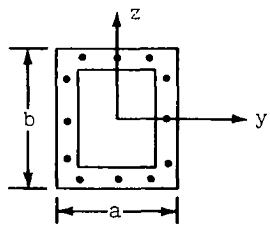
Z - Section
9 Integration Points



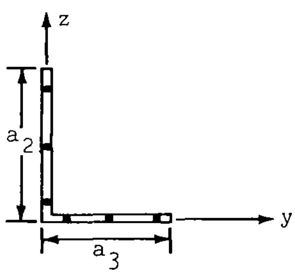
I - Section
9 Integration Points



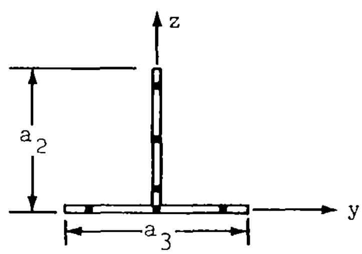
Hollow Circular Section
8 Integration Points



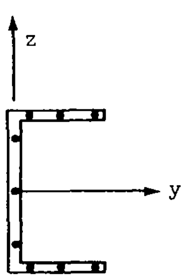
Hollow Rectangular Section
12 Integration Points



L - Section
6 Integration Points



T - Section
6 Integration Points



Channel Section
9 Integration Points

Fig. 13 Beam Sections Available in PLANS

4.4.6 Transformation to Global Coordinates and Offset Specification

Once the element stiffness matrices have been evaluated, it is necessary to assemble each matrix to some common global system. However, since the beam element is geometrically one dimensional, only the orientation of the beam axis is known by specifying the coordinates of the end point nodes. Consequently, the orientation of the cross sectional axes remains undefined. In order to define this orientation an extra node point must be specified as shown in Fig. 14.

This additional node defines a plane that fixes the direction and orientation of the y-axis. The z-axis is perpendicular to this plane. In practice, this node can be part of the structure or it can be specified only for the purpose of defining

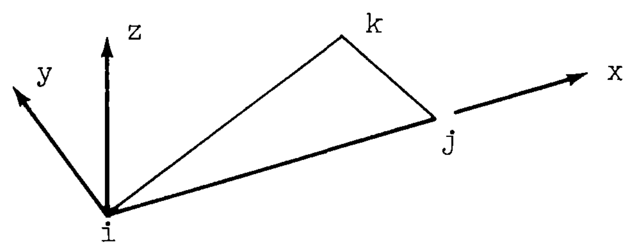


Fig. 14 Beam Node Specification

the cross sectional orientation. If the latter option is chosen all degrees of freedom associated with the node must be fixed. Once this additional point is defined, the direction cosine transformations between the local and global directions are completely determined.

Also accounted for in the transformation between the local and global systems is the offset of the attachment point (point A in Fig. 15) from the beam axis. The appropriate transformation is obtained from the beam kinematic assumptions, Eq. (84), and can be written for the i^{th} node as

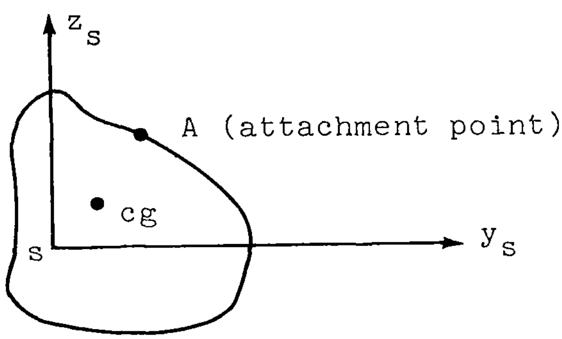


Fig. 15 Definition of Offset Point

$$\begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 & (z_s - z_{cg}) & -(y_s - y_{cg}) \\ 0 & 1 & 0 & z_s & 0 & 0 \\ 0 & 0 & 1 & -y_s & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_{A_i} \\ v_{A_i} \\ w_{A_i} \\ \beta_{x_i} \\ \beta_{y_i} \\ \beta_{z_i} \end{Bmatrix}$$

Geometric nonlinearity is considered in PLANS using an updated coordinate system. In using this method the change in orientation of the beam element must be accounted for during each load step. To this end the change in orientation of each element is calculated as

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix}_1^{i+1} = \begin{bmatrix} \ell_\theta \ell_\beta & \ell_\beta m_\theta & m_\theta \\ -(\ell_\alpha m_\theta + \ell_\theta m_\alpha m_\beta) & \ell_\theta \ell_\alpha - m_\theta m_\beta m_\alpha & \ell_\beta m_\alpha \\ m_\alpha m_\theta - \ell_\alpha \ell_\theta m_\beta & -(\ell_\theta m_\alpha + \ell_\alpha m_\theta m_\beta) & \ell_\alpha \ell_\beta \end{bmatrix} \begin{Bmatrix} x \\ y \\ z \end{Bmatrix}_1^{i}$$

where the coordinates shown are with respect to the beam local axis system and

$$\theta = \tan^{-1}\left(\frac{\Delta v_i - \Delta v_j}{\ell + \Delta u_j - \Delta u_i}\right)$$

$$\beta = \tan^{-1}\left(\frac{\Delta w_j - \Delta w_i}{\sqrt{(\ell + \Delta u_j + u_i)^2 + (\Delta v_j - \Delta v_i)^2}}\right)$$

$$\alpha = \frac{\Delta\beta_{x_j} + \Delta\beta_{x_i}}{2}$$

$$\ell_\theta = \cos\theta \quad , \qquad m_\theta = \sin\theta$$

$$\ell_\beta = \cos\beta \quad , \qquad m_\beta = \sin\beta$$

$$\ell_\alpha = \cos\alpha \quad , \qquad m_\alpha = \sin\alpha$$

Superscripts $i+1$, i represent the current and previous configuration, respectively, and ℓ is the element length in the i^{th} configuration.

4.4.7 Force, Moment, and Stress Calculations

The incremental element nodal forces and moments are obtained in each load increment by making use of the element stiffness matrices related to the local element coordinate system. Thus

$$\{\Delta f\} = \left\{[k^0] + [k^1]\right\}\{\Delta\delta_1\} - [k^*]\{\Delta\epsilon\} \tag{93}$$

Total forces are obtained by summing incremental quantities.

The stresses are evaluated within each increment by substituting the displacement assumptions, Eqs. (84), into the linear portion of the strain displacement relations and then making use of the appropriate stress-strain relations. These equations are evaluated at the Gauss points of each section.

4.4.8 Material Properties

Elastic, ideally plastic, linear strain hardening, or nonlinear strain hardening stress-strain laws may be used with the beam element. The nonlinear hardening is based on a Ramberg-Osgood representation of the actual stress-strain data. The plastic material properties may be changed at the end of each half-cycle of loading.

4.4.9 Loads

The beam element can be loaded by

a) Concentrated forces and moments at each node

b) Linearly varying distributed load in the local y and z directions.

The consistent load vector for the distributed load is obtained from

$$\{f\} = \ell \int_0^1 [N][Q]d\xi \begin{Bmatrix} q_i \\ \hline q_j \end{Bmatrix}$$

where $[N]$ is the shape function for the lateral displacement

$$[N] = \begin{bmatrix} 2\xi^3 - 3\xi^2 + 1 \\ -2\xi^3 + 3\xi^2 \\ \ell(\xi^3 - 2\xi^2 + \xi) \\ \ell(\xi^3 - \xi^2) \end{bmatrix} \qquad (94)$$

$$[Q] = [1 - \xi \;\vdots\; \xi]$$

q_i, q_j are the magnitude of the distributed load at each node in the y or z direction and {f} is the vector of local forces and moments in the x-y or x-z plane.

4.4.10 Comments

- The beam element is available in the OUT-OF-PLANE, OUT-OF-PLANE-MG, and BEND modules of PLANS
- Currently, offsets at both ends of the beam must be the same
- No thermal capability is currently available for the beam element

4.5 Shear Panel Element

4.5.1 Introduction

A warped shear panel has been included in the element library of PLANS. This element is used in practice to model shear webs of wing spars and ribs as well as the sheet area of some fuselage sections. The development of the stiffness properties for this element does not follow the usual consistent development of element properties. The procedure shown herein follows the development as described in Ref. 46.

4.5.2 Stiffness Matrix

The element characteristics for the warped shear panel are developed from the assumption of a distribution of stresses within the element that satisfy equilibrium but do not satisfy strain compatibility, except in the case of a parallelogram. A flexibility coefficient is then computed, using an energy formulation, and a set of equilibrium equations is used to obtain the stiffness matrix. Details involved in obtaining the energy expression can be found in Ref. 47. In order to obtain the stiffness of the element, six equilibrium equations are written for the element in the reference coordinate system shown in Fig. 16.

The forces f_1', f_2', f_3', and f_4' (Fig. 17) are necessary to ensure equilibrium in the z-direction. The next step is to solve for f_1', f_2', f_3', f_4', q_2, q_3, and q_4, in terms of q_1. Since there are six equations and seven unknowns, an additional equation is needed, which is obtained by assuming that the resultant force in the z-direction passes through point v (Fig. 18), and that one-half of this resultant is acting at nodes j and k.

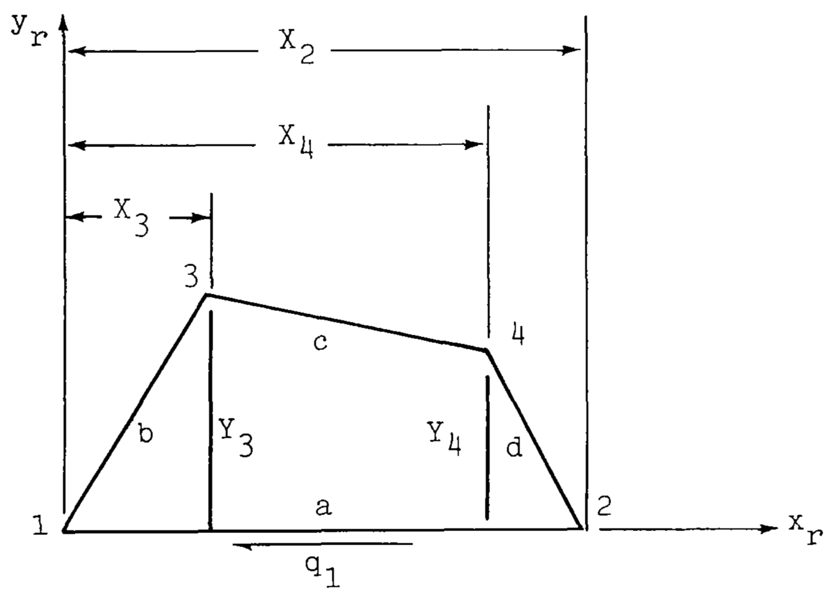


Fig. 16 Projected Quadrilateral

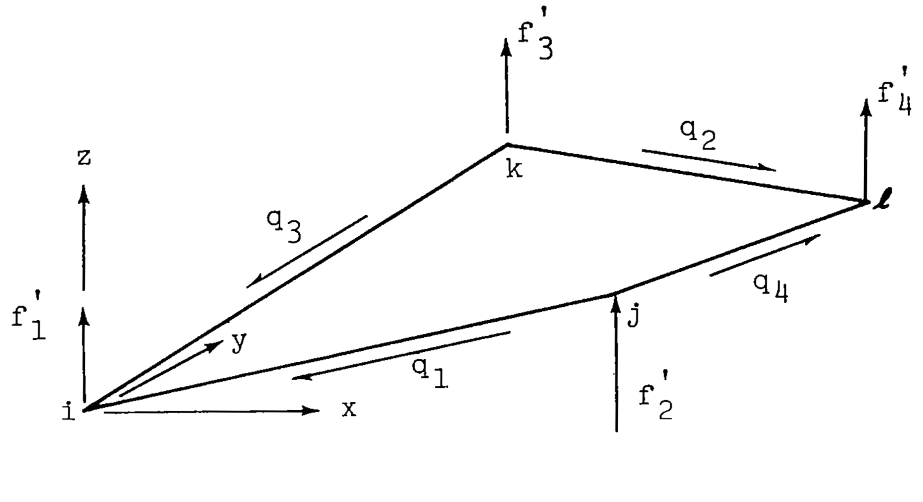


Fig. 17 Equilibrium of Warped Shear Panel

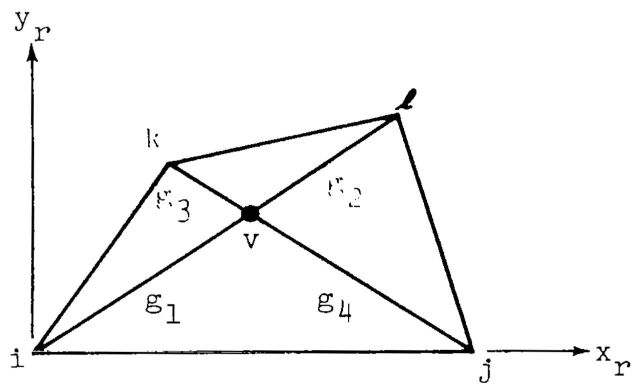


Fig. 18 Additional Geometry for Warped Shear Panel

Taking moments about a line passing through node j and parallel to $i-\ell$

$$f_3' = \frac{1}{2} \frac{g_4}{g_3 + g_4} (f_1' + f_2' + f_3' + f_4') \tag{95}$$

or

$$f_1' + f_2' + (1 - \gamma)f_3' + f_4' = 0 \tag{96}$$

where

$$\gamma = \frac{2(g_3 + g_4)}{g_4} \tag{97}$$

The equilibrium equations and Eq. (96) can be written in the matrix form

$$[EQ]\{f''\} = \{R\}q_1 \tag{98}$$

where $\{f''\} = \{f_1' \ f_2' \ f_3' \ f_4' \ q_2 \ q_3 \ q_4\}$. Solving Eq. (98) we have

$$\{f''\} = [EQ]^{-1}\{R\}q_1 \tag{99}$$

or

$$\{f\} = \{E\}q_1 \tag{100}$$

where $\{f\}$ is obtained by enlarging $\{f''\}$ to include q_1 and $\{E\}$ is obtained by correspondingly enlarging $[EQ]^{-1}\{R\}$ to include the identity $q_1 = q_1$.

The eight applied forces can be considered to correspond to eight degrees of freedom. They are related, as we have seen, by seven equations, six of which are based on static equilibrium and the seventh is an assumed relationship. An additional relationship, based on elastic deformation and corresponding to a single elastic degree of freedom can be introduced. If we regard

q_1 as representing the generalized force corresponding to the elastic degree of freedom and δ_{q_1} is defined as the corresponding generalized displacement, the elastic strain energy may be written in the form

$$U = \tfrac{1}{2}\, \delta_{q_1} q_1 \tag{101}$$

Introducing the relationship

$$\delta_{q_1} = \alpha q_1 \tag{102}$$

where α is a flexibility coefficient, we can write Eq. (101) in the form

$$U = \frac{1}{2\alpha}\, \delta_{q_1}^2 \tag{103}$$

An alternative form for the strain energy is

$$U = \tfrac{1}{2}\, \{f\}^T\{\delta\} \tag{104}$$

where $\{\delta\}$ is a displacement matrix corresponding to $\{f\}$. Substitution of Eq. (100) into Eq. (104) yields

$$U = \tfrac{1}{2}\, \{E\}^T\{\delta\}q_1 \tag{105}$$

Comparing Eqs. (101) and (105), we see that

$$\delta_{q_1} = \{E\}^T\{\delta\} \tag{106}$$

Substitution of Eq. (106) into Eq. (103) yields the form

$$U = \frac{1}{2\alpha}\, \{\delta\}^T\{E\}\{E\}^T\{\delta\} \tag{107}$$

or

$$U = \tfrac{1}{2}\, \{\delta\}^T[k]\{\delta\} \tag{108}$$

where

$$[k] = \frac{1}{\alpha}\{E\}\{E\}^T \tag{109}$$

$[k]$ is seen to be the stiffness matrix in the relationship

$$\{f\} = [k]\{\delta\} \tag{110}$$

The flexibility coefficient α is computed for the projected geometry of the warped quadrilateral on the reference plane defined by the lines 1-4 and 2-3 in Fig. 16. This coefficient gives the relationship between the generalized shear deformation, δ_{q_1}, and the shear flow, q_1, acting along side 1-2, as shown in Fig. 16.

The computation of α for a particular quadrilateral depends on its shape, which can be any of the following four types

- Parallelogram or rectangle
- Trapezoid with side a parallel to side c
- Trapezoid with side b parallel to side d
- General quadrilateral

In general, the expression for α is not easily obtained for trapezoidal and quadrilateral shapes. Approximate expressions have been derived, however, for these shapes by Garvey (Ref. 47) based on the assumption that there are some directions in the panel that are in states of pure shear. This is true for both rectangles and parallelograms, but is only an approximation, for trapezoids and quadrilaterals. Therefore, shapes which vary substantially from a parallelogram introduce errors in the analysis.

4.5.3 Initial Strain Stiffness Matrix

In the presence of initial strains an initial generalized displacement component, $\delta^o_{q_1}$, must be added to Eqs. (102) through (103) so that

$$\delta_{q_1} = \alpha q_1 + \delta^o_{q_1}$$

and

$$\delta_{q_1} = \{E\}^T(\{\delta\} - \{\delta\}^o)$$

where $\{\delta\}^o$ is an initial displacement associated with the forces $\{f\}$ of Eq. (104). This leads to the expression for strain energy

$$U = \tfrac{1}{2}\{\delta\}^T[k]\{\delta\} - \tfrac{1}{2}\{\delta\}^T[E][E]^T\{\delta\}^o + \tfrac{1}{2}\{\delta^o\}^T[E][E]^T\{\delta^o\}$$

Since

$$\{\delta^o_{q_1}\} = \{E\}^T\{\delta^o\}$$

and

$$\{\delta^o_{q_1}\} = \{E\}^T \alpha G\gamma_o$$

where γ_o is an initial strain corresponding to q_1; the complete integral can be written as

$$U = \tfrac{1}{2}\{\delta\}^T[k]\{\delta\}^T - \{\delta\}^T[k]^*\gamma_o - \tfrac{1}{2}\{\delta^o\}^T\{E\}\{E\}^T\{\delta^o\}$$

where the initial strain matrix is

$$[k^*] = \{E\}^T G$$

4.5.4 Comments

- The shear panel is available in the OUT-OF-PLANE module
- The element can currently be elastic only
- The element has no thermal capability
- Force output is in terms of the shear flows and normal forces

4.6 Revolved Triangular Ring Element

4.6.1 Introduction

The revolved ring element is used to describe three dimensional axisymmetric bodies subjected to axisymmetric loads. The element is two dimensional in that there is no variation of stress or strain in the circumferential direction. However, the strains and stresses in the circumferential direction must be considered as they are induced by radial displacements and anisotropy.

The element is an orthotropic triangular ring element similar to that described in the literature (see, for example, Ref. 45). Figure 19 depicts the element along with the principal material directions.

4.6.2 Displacement Assumptions

The element is characterized by a linear displacement assumption, independent of the circumferential direction, i.e.,

$$
\begin{aligned}
u_r &= a_1 + a_2 r + a_3 z \\
u_z &= a_4 + a_5 r + a_6 z \\
u_\theta &= a_7 + a_8 r + a_9 z
\end{aligned}
\tag{111}
$$

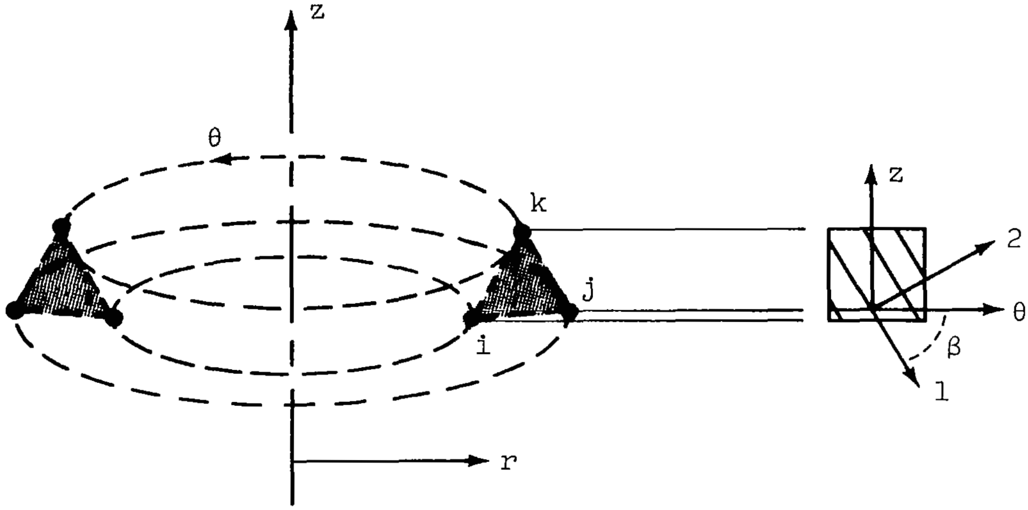


Fig. 19 Revolved Triangular Ring Element and Principal Material Directions

The appropriate strain displacement equations are then

$$
\begin{aligned}
\epsilon_{rr} &= u_{r,r} & \qquad \epsilon_{\theta z} &= u_{\theta,z} \\
\epsilon_{\theta\theta} &= u_r/r & \qquad \epsilon_{rz} &= u_{r,z} + u_{z,r} \\
\epsilon_{zz} &= u_{z,z} & \qquad \epsilon_{r\theta} &= u_{\theta,r} - u_\theta/r
\end{aligned}
\tag{112}
$$

Substituting Eq. (111) into Eq. (112) we arrive at

$$\{\epsilon\} = [B]\{a\} \tag{113}$$

where

$$\{\epsilon\} = \lfloor \epsilon_{\theta\theta} \quad \epsilon_{zz} \quad \epsilon_{rr} \quad \epsilon_{rz} \quad \epsilon_{r\theta} \quad \epsilon_{z\theta} \rfloor^T$$

$$\{a\} = \lfloor a_1 \quad a_2 \quad \cdots \quad a_9 \rfloor^T$$

and

$$[B] = \begin{bmatrix} 1/r & 1 & z/r & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & -1/r & 0 & -z/r \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

From Eq. (111) we can determine $\{a\}$ in terms of the nodal displacements

$$\{a\} = [A]\{u\} \tag{114}$$

where

$$\{u\} = \lfloor u_r^i \quad u_r^j \quad u_r^k \quad u_z^i \quad u_z^j \quad u_z^k \quad u_\theta^i \quad u_\theta^j \quad u_\theta^k \rfloor^T$$

$$[A] = \begin{bmatrix} [M]^{-1} & & \\ & [M]^{-1} & \\ & & [M]^{-1} \end{bmatrix}$$

and

$$[M] = \begin{bmatrix} 1 & r^i & z^i \\ 1 & r^j & z^j \\ 1 & r^k & z^k \end{bmatrix}$$

This leads to an expression of strain in terms of the independent degrees of freedom (nodal displacements),

$$\{\epsilon\} = [W]\{u\} \tag{115}$$

where $[W] = [B][A]$.

4.6.3 Stiffness Matrix

Integrating the elastic strain energy over the whole ring element

$$U = \frac{1}{2} \int_V \{\sigma\}^T \{\epsilon\} dv \tag{116}$$

where $\{\epsilon\}$ can be expressed as the difference between the total strain and initial strain

$$\{\epsilon\} = \{\epsilon_T\} - \{\epsilon_o\}$$

and using the principle of virtual work we arrive at the stiffness matrix

$$[k] = 2\pi \int_V [W]^T [\bar{C}][W] r dr dz \tag{117}$$

where $[\bar{C}]$ is the matrix of elastic constants (see Section 4.6.7).

In order to integrate Eq. (117) we must integrate the following quantities

$$\int \frac{1}{r} drdz = I_1$$

$$\int \frac{z}{r} drdz = I_2 \tag{118}$$

$$\int \frac{z^2}{r} drdz = I_3$$

over the area of the element, as shown in Fig. 20. This integration can be carried out as follows

$$\int_A f(r,z)drdz = \left\{ \int_{r=r_i}^{r=r_k} \int_{z=0}^{z=A_{ik}+B_{ik}r} + \int_{r=r_k}^{r=r_j} \int_{z=0}^{z=A_{kj}+B_{jk}r} - \int_{r=r_i}^{r=r_j} \int_{z=0}^{z=A_{ij}+B_{ij}r} \right\} f(r,z)drdz$$

where

$$A_{mn} = \frac{r_n z_m - r_m z_n}{r_n - r_m} \quad , \quad B_{mn} = \frac{z_n - z_m}{r_n - r_m}$$

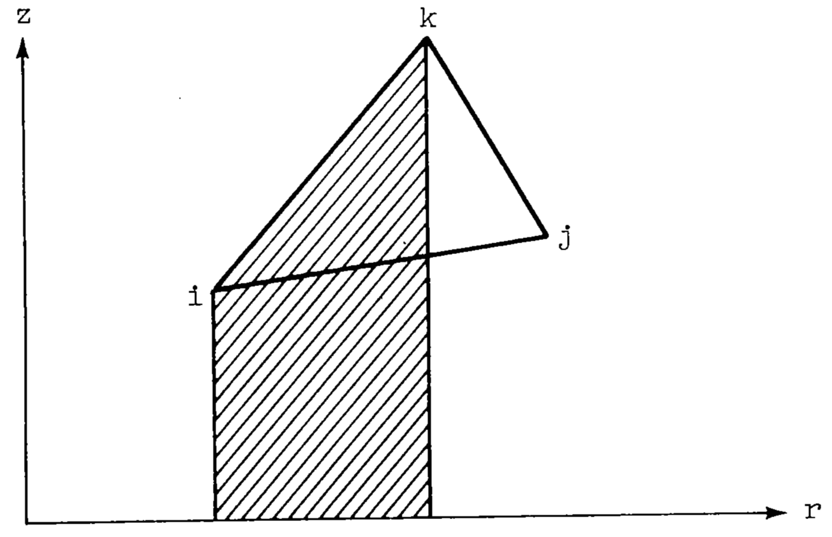


Fig. 20 Integration Limits

Carrying out this integration we arrive at

$$I_1 = (A_{ji} - A_{ik}) \log r_i + (A_{kj} - A_{ji}) \log r_j + (A_{ik} - A_{kj}) \log r_k$$
$$+ B_{ij}(r_i - r_j) + B_{jk}(r_j - r_k) + B_{ik}(r_k - r_i)$$

$$I_2 = \{(A_{ji}^2 - A_{ik}^2) \log r_i + (A_{kj}^2 - A_{ji}^2) \log r_j + (A_{ik}^2 - A_{kj}^2) \log r_k\}/2$$
$$+ A_{ik}B_{ik}(r_k - r_i) + A_{kj}B_{kj}(r_j - r_k) + A_{ji}B_{ji}(r_i - r_j)$$
$$+ \{B_{ik}^2(r_k^2 - r_i^2) + B_{kj}^2(r_j^2 - r_k^2) + B_{ji}^2(r_i^2 - r_j^2)\}/4$$

$$I_3 = \{(A_{ji}^3 - A_{ik}^3) \log r_i + (A_{kj}^3 - A_{ji}^3) \log r_j + (A_{ik}^3 - A_{kj}^3) \log r_k\}/3$$
$$+ A_{ik}^2B_{ik}(r_k - r_i) + A_{kj}^2B_{kj}(r_j - r_k) + A_{ji}^2B_{ji}(r_i - r_j)$$
$$+ \{A_{ik}B_{ik}^2(r_k^2 - r_i^2) + A_{kj}B_{kj}^2(r_j^2 - r_k^2) + A_{ji}B_{ji}^2(r_i^2 - r_j^2)\}/2$$
$$+ \{B_{ik}^3(r_k^3 - r_i^3) + B_{kj}^3(r_j^3 - r_k^3) + B_{ji}^3(r_i^3 - r_j^3)\}/9$$

Two special cases arise. One when $r_m = 0$ which affects the terms involving $(A_{ji}^n - A_{ij}^n) \log r_i$. Using L'Hopital's rule the limit of these terms is zero and can be taken into account by omitting the logarithmic terms from I_1, I_2, and I_3. The second special case arises when $r_i = r_k$ in which case we do not integrate over the area under i-k. This can be accomplished by setting $A_{ik} = B_{ik} = 0$ in I_1, I_2, and I_3. These points are covered in Ref. 45. Note that in the expression for I_1

$$B_{ij}(r_i - r_j) + B_{jk}(r_j - r_k) + B_{ik}(r_k - r_i) = 0$$

except for the case in which $r_i = r_k$ in which case

$$B_{ij}(r_i - r_j) + B_{jk}(r_j - r_k) + B_{ik}(r_k - r_i) = z_i - z_k$$

4.6.4 Initial Strain Matrix

The plastic load vector is given by

$$\{Q\} = 2\pi \int_V [W]^T \, rdrdz \, [\bar{C}]\{\epsilon_o\} = [k^*]\{\epsilon_o\} \tag{119}$$

where $[k^*]$ is the initial strain stiffness matrix and $\{\epsilon_o\}$ is the vector of plastic strains taken to be constant within the element and evaluated at the centroid.

The integration associated with the initial strain matrix, Eq. (119), can be found by evaluating the integrand at the centroid, i.e.,

$$\begin{aligned} \{Q\} &= 2\pi \int_V [W]^T \, rdrdz \, [\bar{C}]\{\epsilon_o\} \\ \{Q\} &= 2\pi FA[\bar{W}]^T[\bar{C}]\{\epsilon_o\} \end{aligned} \tag{120}$$

where A is the cross sectional area, F is the centroidal point, i.e., $F = (r_i + r_j + r_k)/3$ and $[\bar{W}]^T$ is the $[W]^T$ matrix evaluated at the centroid.

4.6.5 Stress Calculation and Evaluation of Plastic Strains

Stresses and plastic strains are calculated at the centroid of the element. All calculated quantities such as stress, plastic strain, shift in yield surface, etc., are output in the principal direction of orthotropy as defined by the angle β (see Fig. 19).

4.6.6 Thermal Stress Calculations

Orthotropic thermal stress calculations are allowed. Different thermal coefficients of expansion may be input in the principal directions of orthotropy. Temperatures are input at nodes. Since a constant thermal strain distribution is assumed within the element, the temperatures at each of the three nodes are averaged. This average value is assumed to hold throughout the element. The thermal load vector is obtained by multiplying the thermal strains by the initial strain stiffness matrix

$$\{\Delta P_{Th}\} = [k^*]\{\Delta \epsilon_{Th}\} \tag{121}$$

where

$$\{\Delta \epsilon_{Th}\} = \begin{Bmatrix} \alpha_1 T \\ \alpha_2 T \\ \alpha_3 T \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

4.6.7 Material Properties

We assume an orthotropic material whose principal directions (1, 2,3) correspond to (1, 2, r) as shown in Fig. 19. We can then express the stress-strain relationship in the principal directions

$$\{\sigma'\} = [C]\{\epsilon'\} \tag{122}$$

where corresponds to the principal direction whose components are $\lfloor 11, 22, rr, r2, r1, 12 \rfloor$ and

$$C_{11} = E_1(1 - \nu_{32}\nu_{23})/\Delta$$
$$C_{12} = C_{21} = E_1(\nu_{12} + \nu_{32}\nu_{13})/\Delta$$
$$C_{13} = C_{31} = E_1(\nu_{13} + \nu_{12}\nu_{23})/\Delta$$
$$C_{22} = E_2(1 - \nu_{13}\nu_{31})/\Delta$$
$$C_{23} = C_{32} = E_2(\nu_{23} + \nu_{13}\nu_{21})/\Delta$$
$$C_{33} = E_3(1 - \nu_{12}\nu_{21})/\Delta$$
$$C_{44} = G_{23}$$
$$C_{55} = G_{13}$$
$$C_{66} = G_{12}$$
$$\Delta = 1 - 2\nu_{12}\nu_{23}\nu_{31} - \nu_{13}\nu_{31} - \nu_{12}\nu_{21} - \nu_{23}\nu_{32}$$

We now express the stresses and strains in the global coordinates as

$$\{\sigma\} = [\bar{C}]\{\epsilon\} \tag{123}$$

where

$$[\bar{C}] = [R]^T[C][R]$$

$$[R] = \begin{bmatrix} \cos^2\beta & \sin^2\beta & 0 & 0 & 0 & -\sin\beta\cos\beta \\ \sin^2\beta & \cos^2\beta & 0 & 0 & 0 & \sin\beta\cos\beta \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\beta & \sin\beta & 0 \\ 0 & 0 & 0 & -\sin\beta & \cos\beta & 0 \\ 2\sin\beta\cos\beta & -2\sin\beta\cos\beta & 0 & 0 & 0 & (\cos^2\beta - \sin^2\beta) \end{bmatrix}$$

In the current version of the REVBY program for strain hardening materials only initially isotropic material behavior can be specified if a plastic analysis is desired. Either a linear or nonlinear hardening law may be chosen. For ideally plastic materials, isotropic or orthotropic plastic material properties may be specified. At the end of each half-cycle the plastic material properties may be changed.

The thermal coefficients of expansion may be specified to be orthotropic in the principal directions. These may not be changed at the end of each half-load cycle.

4.6.8 Loads

Concentrated and Line Loads — External nodal forces $\{F_i\}$ are expressed in force per unit length of circumference except at $r = 0$. The total load applied is therefore $2\pi r\{F_i\}$. At $r = 0$ there must be $F_r = F_\theta \equiv 0$, and F_z represents actual force, in pounds for example.

Surface Tractions — Distributed surface force/unit area can be specified at nodes in the r, z, and θ directions. If we assume a linear variation of traction across the face, i.e., $p = a + b\ell$ (Fig. 21), we arrive at the consistent load vector

$$\begin{Bmatrix} \bar{F}_i \\ \\ \bar{F}_j \end{Bmatrix} = \pi(r_i+r_j)L \begin{bmatrix} \frac{1}{3} & \frac{1}{6} \\ \\ \frac{1}{6} & \frac{1}{3} \end{bmatrix} \begin{Bmatrix} p_i \\ \\ p_j \end{Bmatrix} \tag{124}$$

where p_i corresponds to the traction at node i.

Note: All loads are incremented proportionally from the critical load.

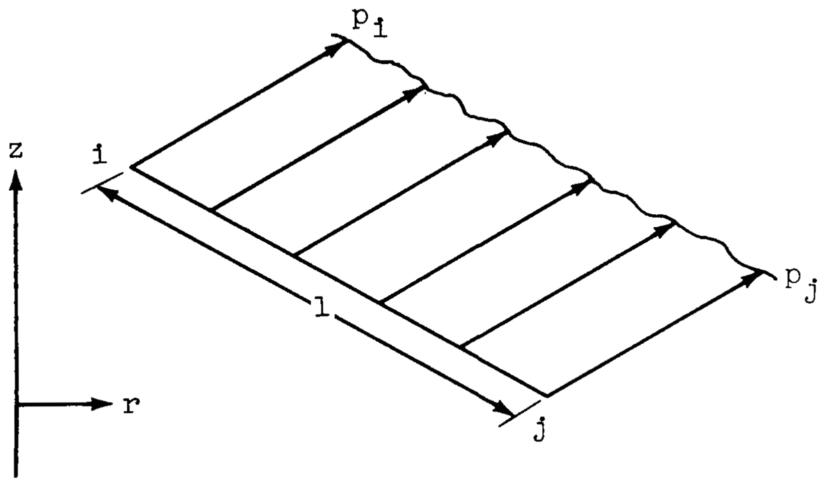


Fig. 21 Distributed Surface Forces

4.6.9 General Comments

- Note that for orthotropic material properties, the R-Z axes and the axes of principal orthotropy must be coincident, i.e., no rotation about the θ-axis is allowed.

- This element is available in the REVBY module.

4.7 Membrane Triangles

4.7.1 Introduction

The membrane triangles described in this section are classified into the following three categories: 1) 3-node constant strain, 2) 6-node linear strain, and 3) 4- and 5-node hybrid elements. The terms constant, linear, and hybrid refer to the strain distributions that exist in the element as a consequence of choosing an assumed displacement variation. A brief description of the elements associated with these three categories follows.

Constant Strain Triangle (CST) – This well-known plane stress membrane element is one of the most widely used elements for the idealization of membrane structures. Its derivation is based on the assumption of a linear distribution for the in-plane displacements u and v, and consequently, leads to a constant strain state within the element. Each vertex is allowed two degrees of freedom (the in-plane displacements u and v) for a total of six degrees of freedom for the element. Consistent with the total strain distribution, the initial strains (plastic strains) are assumed to be constant within each element. Stiffness and initial strain matrices have been developed and successfully used in Refs. 6, 8, and 30.

Linear Strain Triangle (LST)(Fig. 22) – In regions of high strain gradient, the CST triangle is not sufficiently accurate to be used in a plasticity analysis unless a very fine grid is employed. The linear strain triangle (LST) remedies this shortcoming of the CST element. The assumption of a quadratic distribution for the in-plane displacements allows for a linear strain variation within the triangle. Two degrees of freedom at each node (u,v) for each of the six nodes (three vertex and three midside nodes) give this element a total of 12 degrees of freedom. The initial strains are assumed to be constant within each element and are evaluated at the centroid. Both stiffness and initial strain matrices have been developed and successfully used in Refs. 8 and 30.

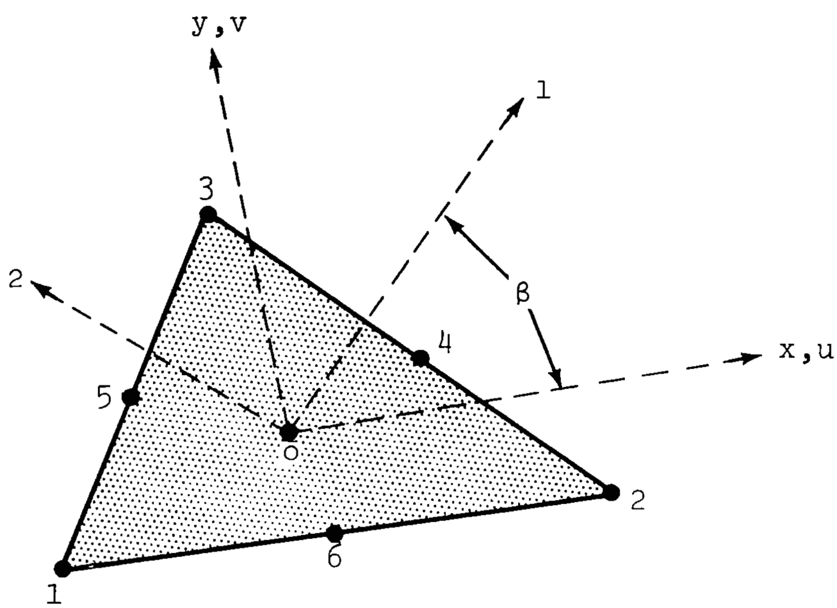


Fig. 22 Linear Strain Triangle

Hybrid Triangles (HST) - In transition regions, i.e., regions in which stresses and strains change from rapidly varying to slowly varying, it becomes convenient and efficient to switch from linear strain triangles to constant strain triangles. This is accomplished by using four-and five-node triangles to maintain compatibility with both the CST and LST elements. These elements together with the CST and LST elements were originally used in Ref. 4 and are referred to as the TRIM 3 through TRIM 6 family. For these mixed formulation hybrid elements, the displacements along edges may vary quadratically or linearly, depending on whether an LST or CST triangle is contiguous to the respective sides. Again, the plastic strain distribution is assumed constant within each element.

4.7.2 Displacement Assumptions

Using a local centroidal coordinate system with the x-axis parallel to side 1-2 shown in Fig. 22, the in-plane displacements u and v may be written in terms of the area coordinates ω_1, ω_2, and ω_3.

$$u = \omega_1(2\omega_1 - 1)u_1 + \omega_2(2\omega_2 - 1)u_2 + \omega_3(2\omega_3 - 1)u_3 + 4\omega_2\omega_3u_4 + 4\omega_3\omega_1u_5 + 4\omega_1\omega_2u_6 \tag{125}$$

or in matrix form as $u = [N]\{u_o\}$ where $[N]$ is a matrix of shape functions representing the assumed displacement field within the element.

The terms u_i $(i = 1\text{-}6)$ refer to nodal displacements at the vertex and midside nodes, the subscript o refers to the nodal displacements, and the area coordinates are written in terms of the local x,y coordinates in the following matrix form

$$\begin{Bmatrix} \omega_1 \\ \omega_2 \\ \omega_3 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} x_2y_3-x_3y_2 & y_2-y_3 & x_3-x_2 \\ x_3y_1-x_1y_3 & y_3-y_1 & x_1-x_3 \\ x_1y_2-x_2y_1 & y_1-y_2 & x_2-x_1 \end{bmatrix}\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} \tag{126}$$

or

$$\begin{Bmatrix} \omega_1 \\ \omega_2 \\ \omega_3 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} 2A_1 & b_1 & a_1 \\ 2A_2 & b_2 & a_2 \\ 2A_3 & b_3 & a_3 \end{bmatrix}\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix}$$

An expression similar to Eq. (125) can be written for the v component of displacement. The particular form of Eq. (125)

represents a quadratic polynomial in the coordinate variables x and y and consequently is associated with the six-node LST element. A linear polynomial representation of the displacements (CST element) is obtained from Eq. (125) if we enforce the following conditions

$$u_4 = \tfrac{1}{2}(u_2+u_3) \quad ; \quad u_5 = \tfrac{1}{2}(u_3+u_1) \quad ; \quad u_6 = \tfrac{1}{2}(u_1+u_2)$$

The hybrid (HST) element is obtained by enforcing any one, or two, of the above conditions.

4.7.3 Stiffness Matrix

Neglecting the effects of large geometry changes, the strain displacement relations are determined by applying the appropriate differential operator on the displacement functions as

$$\begin{Bmatrix} e_x \\ e_y \\ \gamma_{xy} \end{Bmatrix} = [D]\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial x} \end{bmatrix}\begin{Bmatrix} u \\ v \end{Bmatrix} \tag{127}$$

The strain-nodal displacement relations may be represented in matrix form as

$$\{e\} = [D][N]\{u_o\} = [W]\{u_o\} \tag{128}$$

For the LST and HST element the $[W]$ array is a linear combination of the coordinate variables x and y and may be written as the sum of the following terms

$$\underset{3\bullet m}{[W]} = \underset{3\bullet m}{[W_1]} + \underset{3\bullet m}{[W_2]}x + \underset{3\bullet m}{[W_3]}y \tag{129}$$

where m may be 8, 10, or 12. Note that the [W] array will vary with the number of nodes used for the element. For example, for the four-node element, appropriate transformations associated with the displacements removed from two of the three midside nodes must be applied to the full (3x12) [W] array to obtain the correspondingly reduced (3x8) array. Correspondingly, the strain-nodal displacement equation for the CST element is represented by an expression of the form of Eq. (127) with

$$\underset{3\cdot 6}{[W]} = [W_o] \tag{130}$$

The local stiffness matrix for an element of uniform thickness is represented in the following form

$$[k] = h \iint_A [W]^T[E][W]dA \tag{131}$$

where [E] is the array of elastic coefficients relating stress and strain components in local coordinates.

For the LST and HST elements the form for [W] given in Eq. (129) leads to the following expression for the local stiffness matrix

$$\begin{aligned}\underset{m\cdot m}{[k]} &= Ah[W_1]^T[E][W_1] + \frac{Ah}{12}(x_1^2 + x_2^2 + x_3^2)[W_2]^T[E][W_2] \\ &+ \frac{Ah}{12}(x_1y_1 + x_2y_2 + x_3y_3)([W_2]^T[E][W_3] + [W_3]^T[E][W_2]) \\ &+ \frac{Ah}{12}(y_1^2 + y_2^2 + y_3^2)[W_3]^T[E][W_3]\end{aligned} \tag{132}$$

where m = 8, 10 or 12. Similarly, for the CST element the expression for [k] reduces to

$$\underset{6\bullet 6}{[k]} = Ah[W_o]^T[E][W_o] \tag{133}$$

The local stiffness matrices $[k]$ are then transformed to global coordinates by premultiplying them by $[T]^T$ and post-multiplying by $[T]$, where $[T]$ transforms the displacements from local to global quantities.

4.7.4 Geometric Stiffness Matrix

For two dimensional planar problems it is conjectured that large geometry changes (e.g., localized near a crack tip) are basically those involving large rotations. Additionally, the presence of large membrane stresses significantly affects the out-of-plane stiffness of these elements when they are used for built-up structures. This is important in analyzing large deflection and stability problems. On this basis, the inclusion of small strain, large rotation effects (geometric nonlinearity) is effected through the use of the following nonlinear incremental strain-displacement equations

$$\begin{aligned}
\Delta\epsilon_x &= \frac{\partial\Delta u}{\partial x} + \tfrac{1}{2}(\Delta\omega_z)^2 + \tfrac{1}{2}\left(\frac{\partial\Delta w}{\partial x}\right)^2 \\
\Delta\epsilon_y &= \frac{\partial\Delta v}{\partial y} + \tfrac{1}{2}(\Delta\omega_z)^2 + \tfrac{1}{2}\left(\frac{\partial\Delta w}{\partial y}\right)^2 \\
\Delta\gamma_{xy} &= \frac{\partial\Delta u}{\partial y} + \frac{\partial\Delta v}{\partial x} + \frac{\partial\Delta w}{\partial x}\frac{\partial\Delta w}{\partial y}
\end{aligned} \tag{134}$$

where Δ represents an incremental change between two adjacent states of the loading history, and

$$\omega_z = \tfrac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) \tag{135}$$

The displacement w is in the local z-direction. The displacement assumption used for w is the same as that used for u and v in corresponding elements.

The nonlinear terms of Eq. (134) give rise to the geometric (or initial stress) stiffness matrix. Its local form may be represented as follows for an element of uniform thickness

$$[k^1] = \iint_A [\Omega]^T[\Sigma][\Omega]dA \tag{136}$$

The matrix $[\Omega]$ is used to express the rotations in terms of nodal displacements, e.g.,

$$\begin{Bmatrix} \Delta w_{,x} \\ \Delta w_{,y} \\ \Delta\omega_z \end{Bmatrix} = [\Omega]\{\Delta u_o\} \tag{137}$$

where $\{\Delta u_o\}$ now includes the w terms and $[\Sigma]$ is, using a convected coordinate (updated geometry) approach for the solutionof geometric nonlinear problems merely an array of stress resultants N_x, N_y, N_{xy}, for each element (or node).

Centroidal values of the stress resultants are used in all order elements. The local geometric stiffness matrices of Eq. (136) are then transformed into the global system by premultiplying them by $[T]^T$ and postmultiplying them by $[T]$.

4.7.5 Initial Strain Stiffness Matrix

The treatment of plasticity within the PLANS system requires the incorporation of two basic assumptions. The first is, in determining the effects of plastic flow within a body, plastic strains are interpreted as initial strains; the second is that total strains (or their increments) may be decomposed into elastic and plastic components. The local initial strain matrix that appears in the equilibrium equations may be written in terms of previous variables as

$$[k^*] = h \iint_A [W]^T [E][N^*] dA \tag{138}$$

where $[N^*]$ represents the matrix of shape functions used to describe the assumed plastic strain variation within an element. If we choose to consider the plastic strain state at the centroid of the element as representative for the element (regardless of the total strain variation) then Eq. (138) may be written as

$$\underset{m\bullet 3}{[k^*]} = Ah[W_1][E] \tag{139}$$

where $m = 8$, 10, or 12 for the LST and HST elements, and

$$\underset{6\bullet 3}{[k^*]} = Ah[W_o][E] \tag{140}$$

for the CST element.

Plastic Load Vector — The product of the initial strain matrix and the vector of plastic strains represents the "effective" plastic load for the structure. This load is applied in addition to the prescribed set of mechanical loads to determine the displacement state. The aforementioned product is determined at the element level; that is, rather than assembling an initial strain coefficient matrix for the entire structure and multiplying by the vector of plastic strains, the vector product of the element initial strain matrix and plastic strains for the element is assembled for the entire structure. Thus, the local plastic load vector may be represented in the following form

$$\{Q\} = \sum_{i=1}^{N} \underset{m \bullet 3}{[k^*]_i} \begin{Bmatrix} \epsilon_x^P \\ \epsilon_y^P \\ \gamma_{xy}^P \end{Bmatrix}_i \tag{141}$$

where $m = 6, 8, 10,$ or 12, N = number of plastic elements, and ϵ_x^P, ϵ_y^P, and γ_{xy}^P represent plastic strain components. This vector is then premultiplied by $[T]^T$ to transform it to global coordinates.

4.7.6 Stress Calculations

The total strains for each element are evaluated at the centroid using the strain-displacement relations previously discussed. For small deformations the linear relation becomes

$$\{e\} = \underset{3 \bullet m}{[W_1]}\{u_o\} \tag{142}$$

where $m = 8, 10,$ or 12 for the LST and HST elements, and

$$\{e\} = \underset{3 \bullet 6}{[W_o]}\{u_o\} \tag{143}$$

for the CST element. Once again, note that $[W_1]$ will vary depending on the value of m.

The nonlinear contribution to the strains, as expressed in Eq. (134) requires the $[\Omega]$ array to be evaluated at the centroid. Stresses and plastic strains are also evaluated at the centroid for LST, HST, and CST elements. Although the stresses in an LST or HST element are not uniform, but vary from node to node within an element, the centroidal value is used to represent the stress state of the element. The plastic strains, on the other hand, are a priori assumed to be constant within the element regardless of the element type (e.g., LST, HST, or CST). It should be noted that using a constant centroidal value of stress for the LST and HST elements does not degrade the stiffness influence coefficients for these elements. It does, however, influence the nonlinear response of the structure in that plastic strains are determined from these stresses. The inaccuracies associated with using centroidal stresses is most pronounced in regions of rapid stress gradients.

4.7.7 Thermal Stress Calculations

Nodal temperature input is converted to average element temperatures and subsequently to centroidal thermal strains for LST, HST, and CST elements. Orthotropic thermal coefficients of expansion may be specified in the principal material directions of orthotropy. The thermal load vector is determined in the same manner as the plastic load vector, e.g., as the product of the initial strain matrices and the vector of thermal strains for the element and summed over all the elements of the structure.

4.7.8 Material Properties

All elastic material properties are constant within the element and are assumed to be temperature independent. Orthotropic properties may be specified by defining the necessary material constants and the direction of the principal axis of orthotropy, β, for each element. In general the stress-strain relation may be written in an arbitrarily rotated, orthogonal, 1-2 plane (Fig. 22) as

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & 0 \\ C_{21} & C_{22} & 0 \\ 0 & 0 & C_{33} \end{bmatrix} \begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \gamma_{12} \end{Bmatrix} \tag{144}$$

where

$$C_{11} = E_1/(1 - \nu_{21}\nu_{12})$$

$$C_{12} = \nu_{12}C_{11}$$

$$C_{22} = E_2/(1 - \nu_{21}\nu_{12})$$

$$C_{21} = \nu_{21}C_{22}$$

$$C_{33} = G_{12}$$

and $C_{11}C_{22} - C_{12}^2 > 0$, E_1, E_2, and $G_{12} > 0$ for a positive definite stiffness.

Thermal coefficients of expansion in the 1 and 2 directions may be specified independently.

The von Mises yield criterion is used to define the limits of elastic behavior for isotropic materials. Hill's yield condition is used for materials having different yield stresses in the two orthogonal global directions. This yield criterion for plane stress is written as

$$f = (G+H)\sigma_{11}^2 + (F+H)\sigma_{22}^2 - 2H\sigma_{11}\sigma_{22} + 2N\tau_{12}^2 = 1 \tag{145}$$

where

$$G+H = 1/X^2$$

$$F+H = 1/Y^2$$

$$2H = 1/X^2 + 1/Y^2 - 1/Z^2$$

$$2N = 1/T^2$$

and X,Y,Z represent the normal tensile yield stresses in the 1, 2, and 3 directions, respectively, and T is the yield stress in shear in the 1-2 plane. Hill's condition, Eq. (134), reduces to the von Mises yield condition for isotropic yield stresses. Furthermore, the following condition must be enforced to ensure the convexity of the orthotropic yield surface with respect to the origin in stress space

$$\frac{1}{X^2Y^2} - \frac{1}{4}\left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)^2 > 0 \tag{146}$$

Three options are available to describe the nonlinear material behavior: 1) elastic-ideally plastic, 2) elastic-linear hardening, and 3) elastic-nonlinear hardening. For the first option only the yield stresses, X,Y,Z, and T need be specified. For hardening behavior the kinematic hardening theory as proposed by Prager (Ref. 42) and modified by Ziegler (Ref. 37) is used to describe the subsequent hardening behavior of the material beyond initial yielding. For linear hardening the slope of the tensile stress-plastic strain curve is specified (tangent modulus). If nonlinear hardening is desired the Ramberg-Osgood parameters (Ref. 38) must be specified (see Section 3.4 for further discussion). For orthotropic kinematic hardening the Ramberg-Osgood parameters must be specified for each of the three stress components.

4.7.9 Loads

The following mechanical loads may be applied to the membrane elements:

- Concentrated forces and moments applied at specified nodes in the global directions in the units of force or force times length.

- Distributed edge loads in the plane of the element. These are assumed to vary linearly from node to adjacent node along the edge on which the load is applied. The magnitudes of the loads are specified in the global directions at each pair of adjacent nodes.

The consistent load vector, $\{P\}$, associated with the distributed load case is determined from the shape function, $[N]$, Eq. (125),

$$\{P\} = \int_A [N]^T \{p\} dA \tag{147}$$

4.7.10 General Comments

- The same assumed constant plastic strain distribution is used for all of the elements of the membrane type (CST, HST, and LST). Although this distribution is compatible with the total strains in the CST elements only, it should not introduce a significant degradation of the accuracy associated with the HST and LST elements.

- Centroidal values of stresses and strains are used for all the elements.

- Although the situation illustrated in Fig. 23 is allowed (no error message triggered) it is not recommended. The displacement field along the common edge A-B from each of the two adjacent elements will not be compatible.

- The applicable modules for these elements are BEND, OUT-OF-PLANE, and OUT-OF-PLANE-MG. They are also available in a special fracture module called FAST as well as in preliminary versions of OUT-OF-PLANE designated as PLANE.

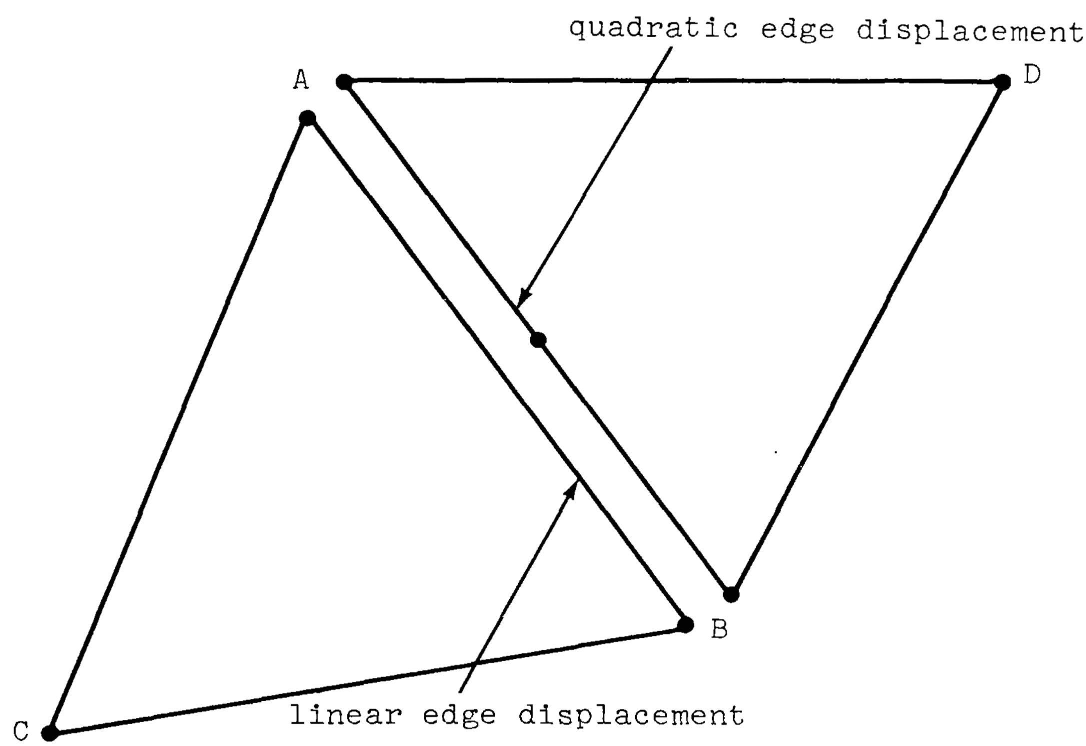


Fig. 23 Incompatible Edge Displacements Along A-B

4.8 Triangular Flat Plate Element - Bending and Membrane

4.8.1 Introduction

This element is used for analyzing thin-walled structures where local bending as well as membrane effects are important. The element is of constant thickness, triangular planform, and has 12 degrees of freedom at each node (see Fig. 24). It is obtained by superposing the Quadratic Strain Membrane Triangle of Felippa (Ref. 16) and Tocher and Hartz (Ref. 48), with the

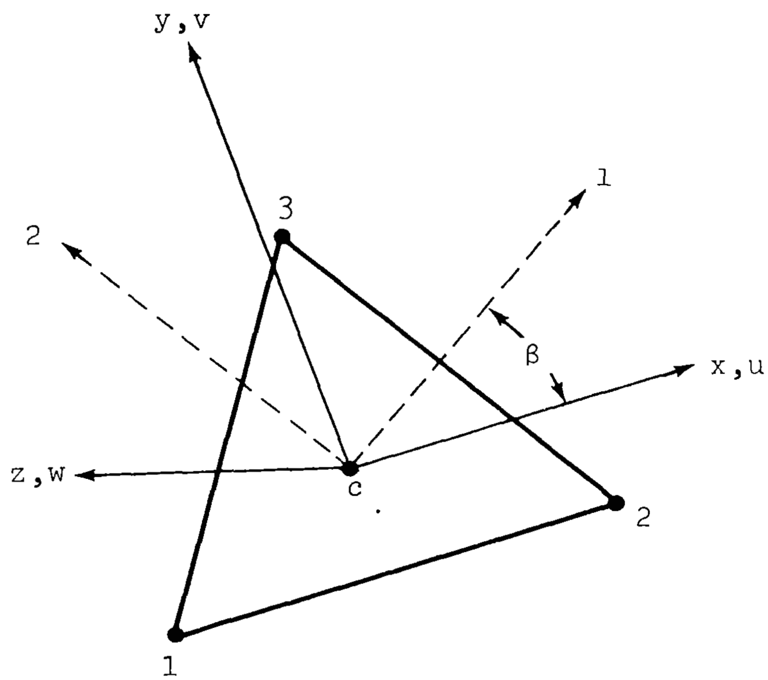


Fig. 24 Triangular Plate Element - Bending and Membrane

quintic displacement bending triangle of Bell (Ref. 47) and Cowper et al. (Ref. 50). The local coordinate system is defined at the centroid of the element. The local x-axis is parallel to the 1-2 edge. The local y-axis is perpendicular to the x-axis and in the plane of the element. The local z-axis is perpendicular to both of these axes.

4.8.2 Displacement Assumptions

$$\begin{aligned} u &= a_o + a_1 x + a_2 y + \ldots + a_{10} y^3 \\ v &= b_o + b_1 x + b_2 y + \ldots + b_{10} y^3 \\ w &= c_o + c_1 x + c_2 y + \ldots + c_{10} y^3 + \ldots c_{21} y^5 \end{aligned} \tag{148}$$

The in-plane displacements u,v are complete cubic polynomials. The 20 undetermined coefficients (a_i and b_i) are related to nodal degrees of freedom through the matrix $[A_m]^{-1}$. These nodal quantities are u_i, v_i, $u_{,x_i}$, $v_{,x_i}$, $u_{,y_i}$, $v_{,y_i}$ where $i = 1, 2, 3$, and u_c, v_c, the centroidal displacements

$$\{a_m\} = [A_m]^{-1}\{d_m\} \tag{149}$$

The 20 in-plane degrees of freedom associated with u and v are reduced to 18 in-plane vertex nodal degrees of freedom by Gaussian elimination (equivalent to static condensation) of the centroidal degrees of freedom u_c, v_c. The remaining nodal degrees of freedom are u_i, v_i, θ_{z_i}, ϵ_{x_i}, ϵ_{y_i}, ϵ_{xy_i}, where θ_{z_i} is the local rotation about the z-axis, $\theta_{z_i} = \frac{1}{2}(-\partial u_i/\partial y + \partial v_i/\partial x)$, and $\epsilon_{x_i} = \partial u_i/\partial x$, $\epsilon_{y_i} = \partial v_i/\partial y$, $\epsilon_{xy_i} = \frac{1}{2}(\partial u_i/\partial y + \partial v_i/\partial x)$ are the tensor components of the membrane strains.

The out-of-plane displacement w is a complete quintic polynomial in x,y. The 21 coefficients (c_i) are related to the vertex nodal degrees of freedom w_i, $w_{,x_i}$, $w_{,y_i}$, $w_{,xx_i}$, $w_{,yy_i}$, $w_{,xy_i}$, and the normal midside nodal rotations $w_{,n_i}$ through the matrix $[A_b]^{-1}$

$$\{a_b\} = [A_b]^{-1}\{d_b\} \tag{150}$$

The 21 degrees of freedom are reduced to 18 by requiring that the normal slopes vary cubically along an edge (see Ref. 49), thus eliminating the three normal slopes at the midside nodes. The local bending degrees of freedom remaining at each vertex node are w_i, $\theta_{x_i} = w_{,y_i}$, $\theta_{y_i} = -w_{,x_i}$, and $\kappa_{x_i} = w_{,xx_i}$, $\kappa_{y_i} = w_{,yy_i}$, $\kappa_{xy_i} = w_{,xy_i}$.

The bending and membrane stiffnesses are then superposed to obtain a 36 degrees of freedom element with the following 12 local degrees of freedom at each node u, v, w, θ_x, θ_y, θ_z, ϵ_x, ϵ_y, ϵ_{xy}, κ_x, κ_y, κ_{xy}.

4.8.3 Formation of Stiffness Matrix

The membrane and bending stiffness matrices may be written as

$$\begin{aligned}
[k_m] &= h[A_m]^{-T} \int_A [W_m]^T[E][W_m]dA[A_m]^{-1} \\
[k_b] &= \frac{h^3}{12} [C]^T[A_b]^{-T} \int_A [W_b]^T[E][W_b]dA[A_b]^{-1}[C]
\end{aligned} \tag{151}$$

Here h is the plate thickness; $[W_m]$ and $[W_b]$ are the matrices relating the strains to nodal parameters as

$$\{\epsilon_m\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{Bmatrix} = [W_m]\{a_m\}$$

$$\{k_b\} = \begin{Bmatrix} \kappa_x \\ \kappa_y \\ 2\kappa_{xy} \end{Bmatrix} = -z \begin{Bmatrix} w_{,xx} \\ w_{,yy} \\ 2w_{,xy} \end{Bmatrix} = -z[W_b]\{a_b\} \tag{152}$$

and $[C]$ is the condensation matrix for the bending component. $[E]$, the matrix of elastic constants relating stress to strain, is discussed more fully in the section on material properties. After formation of the full 20 degrees of freedom membrane stiffness matrix it is reduced and reordered. The area integration for bending and membrane is performed using Gauss-Legendre integration of fourth order.

4.8.4 Formation of Geometric Stiffness Matrix

The geometric stiffness matrix was developed to be used with an updated coordinate system approach (Ref. 26) for solving geometric nonlinear problems. Small-strain moderate rotation assumptions were made to arrive at the strain-displacement equations used. These are

$$\epsilon_x^m = e_x + \tfrac{1}{2}(\omega_y^2 + \omega_z^2) = \frac{\partial u}{\partial x} + \tfrac{1}{2}\left(\frac{\partial w}{\partial x}\right)^2 + \tfrac{1}{8}(v_{,x} - u_{,y})^2$$

$$\epsilon_y^m = e_y + \tfrac{1}{2}(\omega_x^2 + \omega_z^2) = \frac{\partial v}{\partial y} + \tfrac{1}{2}\left(\frac{\partial w}{\partial y}\right)^2 + \tfrac{1}{8}(v_{,x} - u_{,y})^2 \qquad (153)$$

$$\gamma_{xy}^m = e_{xy} - \omega_x\omega_y = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} + \frac{\partial w}{\partial x}\frac{\partial w}{\partial y}$$

$$\epsilon_x^b = -zw_{,xx} \quad , \quad \epsilon_y^b = -zw_{,yy} \quad , \quad \gamma_{xy}^b = -2zw_{,xy}$$

where ω_x, ω_y, and ω_z are infinitesimal rotations.

The component of the geometric stiffness that affects the bending stiffness is written as

$$[k_{Gb}] = [C]^T[A_b]^{-T} \int_A [\hat{W}_b]^T[N][\hat{W}_b]\, dA[A_b]^{-1}[C] \qquad (154)$$

Here

$$[N] = \begin{bmatrix} N_x & N_{xy} \\ N_{xy} & N_y \end{bmatrix}$$

and

$$\begin{Bmatrix} w_{,x} \\ w_{,y} \end{Bmatrix} = [\hat{W}_b]\{a_b\}$$

where $[\hat{W}_b]$ relates the slopes to the nodal parameters a_b. The geometric stiffness that affects the in-plane stiffness is

$$[k_{Gm}] = (N_x+N_y)[A_m]^{-T} \int_A [\hat{W}_m]^T[\hat{W}_m]dA[A_m]^{-1} \qquad (155)$$

where $\omega_z = [\hat{W}_m]\{a_m\}$ and $[\hat{W}_m]$ is a row vector relating the rotation ω_z to the nodal parameters $\{a_m\}$.

These matrices are added to the conventional stiffness matrices and reordered and reduced in the manner discussed in the description of the conventional stiffness matrices. For problems involving geometric nonlinearity all stiffness matrix integrations are performed using fifth order Gauss-Legendre integration over the triangular regions. The values of N_x, N_y, and N_{xy} used to form the geometric stiffnesses are obtained by averaging respective nodal quantities from each of the three nodes.

4.8.5 Formation of Initial Strain Matrix (Plastic Load Vector)

The membrane and bending initial strain matrices are written as (Ref. 26)

$$
\begin{aligned}
[k_m^*] &= [A_m]^{-T} \int_A [W_m]^T [E][\bar{C}]dA \\
[k_b^*] &= [C]^T [A_b]^{-T} \int_A [W_b]^T [E][\bar{C}]dA
\end{aligned}
\tag{156}
$$

The initial plastic strains are assumed to vary linearly from node to node within the element and have an arbitrary variation through the thickness at the nodes, i.e.,

$$\{\epsilon^P(x,y,z)\} = [\bar{C}(x,y)]\{\epsilon_i^P(z)\} \tag{157}$$

where $[\bar{C}(x,y)]$ is a linear function matrix in (x,y) relating the plastic strains in the element to the nodal plastic strains. The area integration for the bending and membrane components of the initial strain stiffness matrices is performed using Gauss-Legendre integration of fourth order. The membrane component of the initial strain matrix is condensed from 20 to 18 degrees of freedom using Gaussian elimination.

The plastic load vector $\{\Delta Q\}$ is the product of the initial strain matrix and the vector of initial strains or, for plastic analysis of plates (and shells), inelastic forces and moments,

$$\{\Delta Q\} = [k^*]\{\Delta\epsilon\} \tag{158}$$

where

$$\{\Delta\epsilon\} = \left\{\begin{array}{c} \begin{bmatrix} \int \Delta\epsilon_x dz \\ \int \Delta\epsilon_y dz \\ \int \Delta\gamma_{xy} dz \\ \int \Delta\epsilon_x zdz \\ \int \Delta\epsilon_y zdz \\ \int \Delta\gamma^p_{xy} zdz \end{bmatrix}_i \\ ---- \\ \begin{bmatrix} \int \Delta\epsilon_x dz \\ \vdots \\ \int \Delta\gamma^p_{xy} zdz \end{bmatrix}_j \\ ---- \\ \begin{bmatrix} \int \Delta\epsilon_x dz \\ \vdots \\ \int \Delta\gamma^p_{xy} zdz \end{bmatrix}_k \end{array}\right\}$$

The plastic strains are evaluated at the nodes of the triangle and integrated through the thickness using Simpson's rule.

Up to 20 layers (21 points) are allowed. The number of layers must be an even number. If the number of layers is not specified then the default of 10 layers is used. This has been found to be sufficient for most problems.

4.8.6 Stress Calculations and Evaluation of Plastic Strains

Stresses are calculated at each of the three nodes of the triangle. There is no averaging of membrane strains or curvatures from adjacent triangles. Since the orthotropic material properties are given in principal directions of anisotropy, all calculated stresses are in the element material coordinate system defined by the angle β (see Fig. 24). These directions must also be used in conjunction with Hill's orthotropic yield criterion. Stresses are also calculated at each integration point through the thickness and will be printed out at these points if requested. Otherwise just top and bottom surface stresses and strains are output.

4.8.7 Thermal Stress Calculations

Orthotropic thermal stress calculations are allowed. Different thermal coefficients of expansion in the principal directions of orthotropy may be specified. A parabolic temperature distribution through the thickness is assumed. Temperatures at the top, bottom, and middle surface through the thickness at each node are input. Like the plastic strains, temperatures are assumed to vary linearly from node to node. Temperatures at each layer through the thickness are determined from the assumed parabolic distribution. The thermal load vector is obtained by multiplying the thermal strains by the initial strain stiffness matrix

$$\{\Delta P_{Th}\} = [k^*]\{\Delta\epsilon_{Th}\}$$

where

$$\{\Delta\epsilon_{Th}\}_i = \begin{Bmatrix} \int \alpha_1 T dz \\ \int \alpha_2 T dz \\ 0 \\ - - - - \\ \int \alpha_1 T z dz \\ \int \alpha_2 T z dz \\ 0 \end{Bmatrix}_i \tag{159}$$

The integrations are evaluated in closed form based on the parabolic temperature distribution.

4.8.8 Material Properties

All material properties are constant within the element and are assumed temperature independent. Elastic properties are assumed to be orthotropic with the direction of the principal axis of orthotropy, β, specified for each element

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} \frac{E_1}{1-\nu_{21}\nu_{12}} & \frac{\nu_{12}E_1}{1-\nu_{21}\nu_{12}} & 0 \\ \frac{\nu_{21}E_2}{1-\nu_{21}\nu_{12}} & \frac{E_2}{1-\nu_{21}\nu_{12}} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} C_{11} & C_{12} & 0 \\ C_{21} & C_{22} & 0 \\ 0 & 0 & C_{33} \end{bmatrix} \{\epsilon\} \tag{160}$$

where $C_{11}C_{22}-C_{12}^2 > 0$, $E_1, E_2, G_{12} > 0$ for positive definite stiffness. Thermal coefficients of expansion in the 1 and 2 directions may also be specified independently.

Plastic material properties may be assumed to be orthotropic with Hill's yield criterion for plane stress used for initial yielding. This yield criterion is written as

$$f = (G+H)\sigma_1^2 + (F+H)\sigma_2^2 - 2H\sigma_1\sigma_2 + 2N\tau_{12}^2 = 1 \tag{161}$$

where

$$G+H = \frac{1}{X^2}$$

$$F+H = \frac{1}{Y^2}$$

$$2H = \frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}$$

$$2N = \frac{1}{T^2}$$

and X,Y, and Z represent the normal tensile yield stresses in the 1,2, and 3 directions, respectively, and T is the yield stress in shear in the 1-2 plane. The following additional stability criterion must be satisfied by the yield stresses for Hill's equations to be used

$$\frac{1}{X^2}\frac{1}{Y^2} - \frac{1}{4}\left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)^2 > 0 \tag{162}$$

Kinematic hardening theory as proposed by Prager (Ref. 42) and modified by Ziegler (Ref. 37) is used to describe the subsequent hardening behavior of the material.

Three different plasticity options are available: 1) elastic, ideally plastic, 2) elastic linear strain hardening, and 3) elastic nonlinear strain hardening using a Ramberg-Osgood power law representation of the stress-strain law. For ideally plastic materials only the yield stresses X,Y,Z,T need to be specified. For linear strain hardening the slopes of the plastic portion of the stress-strain curves (tangent moduli) must be input for each of the three stress components. If nonlinear hardening is desired then two Ramberg-Osgood shape parameters (see Section 3 for further discussion) for each stress component must be specified.

Different stress-strain components may have different hardening laws, e.g., σ_x-ϵ_x nonlinear hardening, σ_y-ϵ_y linear hardening, and τ_{xy}-γ_{xy} nonlinear hardening. However, if one component is elastic-ideally plastic all must be. Additionally, at the end of each half-cycle of loading, the plastic material properties may be changed. Elastic properties may not be changed at the end of each half-cycle.

4.8.9 Loadings

The following mechanical loads may be applied to the plate element:

- Concentrated forces and moments
- Distributed edge loads perpendicular to and in the element plane
- Distributed surface loads perpendicular to and in the plane of the element.

Concentrated Forces and Moments — These are applied at specified nodes in the global directions.

Edge Loads — These are assumed to vary linearly from node to adjacent node along the edge on which they are applied. They are applied in the element local coordinate system. The three components of the distributed edge load are specified at the i^{th} and j^{th} node of the edge. Edge 1 is defined to be the side of the element connecting nodes 1 and 2; edge 2 is defined to be the side connecting nodes 2 and 3; and edge 3 is defined to be the side connecting nodes 3 and 1 (Fig. 24). Applicable members with the same distribution along the respective edges are then specified. Consistent load vectors for these loadings are calculated in the element routines

$$\{P_m\} = [A_m]^{-T} \int_S [\bar{N}_m]^T \{\bar{P}_m\} dS \; ; \begin{Bmatrix} u \\ v \end{Bmatrix} = [\bar{N}_m]\{a_m\}$$

$$\{P_b\} = [C]^T [A_b]^{-T} \int_S [\bar{N}_b]^T \{\bar{P}_b\} dS \; ; \{w\} = [\bar{N}_b]\{u_b\} \tag{163}$$

These are then reordered and reduced as described in the initial strain section.

Distributed Surface Loads – The three components of the surface loads are assumed to vary linearly from node to node in the plane of the element. They are applied in the element local coordinate system perpendicular to and in the x,y directions. The three components of the surface load p_x, p_y, p_z are specified at the 1,2,3 nodes of the element. Applicable members with the same values of components of the respective nodes are then specified. Consistent load vectors are calculated in the element routines. These are determined from the following relations.

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = [\bar{N}_m]\{a_m\}$$

$$\{w\} = [\bar{N}_b]\{a_b\}$$

The load vectors can be written as

$$\{P_m\} = [A_m]^{-T} \int_A [\bar{N}_m]^T\{\tilde{p}_m\}dA$$

$$\{P_b\} = [C]^T[A_b]^{-T} \int_A [\bar{N}_b]^T\{\tilde{p}_b\}dA \tag{164}$$

These are then reordered and reduced as in the initial strain section.

4.8.10 Equilibrium Correction

We may write the equilibrium correction terms for the plate element as (Ref. 36)

$$\{R\} = \{P_i\} - \int_V [W]^T\{\sigma_o\}dV \tag{165}$$

where

$$[W] = \left[\begin{array}{c|c} W_m & 0 \\ \hline 0 & W_b \end{array}\right]$$

Assuming a linear stress variation from node to node this can be rewritten as

$$\{R\} = \{P_i\} - \int_A \left[\begin{array}{c|c} [A_m]^{-T}[W_m][\bar{C}] & 0 \\ \hline 0 & -[C]^T[A_b]^{-T}[W_b][\bar{C}] \end{array}\right] dA \begin{Bmatrix} N_o \\ M_o \end{Bmatrix} \tag{166}$$

The $\{P_i\}$ component is formed in the load vector formation and merely consists of multiplying the consistent load vector by the multiplicative scalar appropriate to the current load value. The second component is virtually identical to the initial strain matrix and is formed at the same time. It is multiplied by the vector of stress and moment resultants. The latter is obtained by numerically integrating the stresses through the thickness using Simpson's rule.

4.8.11 General Comments

- In general, displacements and rotations between elements along an edge will not be compatible if the adjacent elements are not in the same plane. This is due to the assumption of a quintic polynomial representation for the out-of-plane displacement w and a cubic representation for u and v. If the structure is planar then displacements and rotations will be compatible and membrane strains and curvatures will be identical in elements with common nodes.

- The displacements and rotations are assembled in the global system. Membrane strain and curvature degrees of freedom are assembled in a "local global system." The local coordinate system of the first element specified that contains a node becomes the reference system for that node. For this reason the out-of-plane angles between elements should not be large and each grouping of elements about a node should be a "shallow shell." If this is not so, for example, at a node on the corner of a box, the membrane strains and curvatures should not be assembled with those of an element with which there is a large angle. To avoid linking all of the degrees of freedom of the two nodes, multiple definition of a point with two node numbers can be made and the same coordinates assigned to the two nodes. Multipoint constraints can then be used to selectively link displacement and rotation degrees of freedom to ensure continuity of these quantities.

- This element is available in the BEND module.

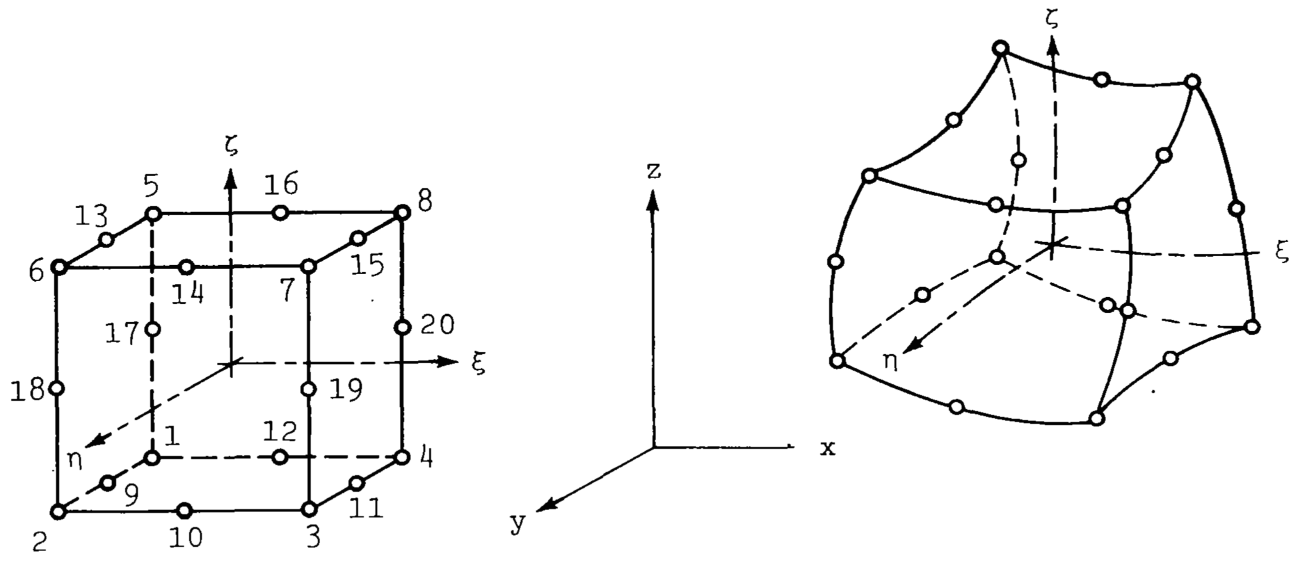
4.9 Isoparametric Hexahedron

4.9.1 Introduction

A review of the present state of isoparametric elements is presented in Ref. 51. These elements are characterized by a mapping procedure used to map a simple shape in the local or natural system into a curvilinear shape (actual shape) in the global system. For the three dimensional solid element presented here, a cube in local coordinates is mapped into a hexahedron in global coordinates (Fig. 25). The mapping function can be written in the form

$$\begin{Bmatrix} x \\ y \\ z \end{Bmatrix} = \sum_i N_i'(\xi,\zeta,\eta) \begin{Bmatrix} x_i \\ y_i \\ z_i \end{Bmatrix} \qquad (167)$$

where x, y, and z represent the global coordinates, ξ,ζ, and η represent the local coordinates, and x_i represent the



a) Local Coordinates b) Global Coordinates

Fig. 25 Isoparametric Hexahedron

nodal coordinates in the global system. The mapping function, or shape functions, N_i', have the property of being unity at node i and zero at all other nodes.

4.9.2 Displacement Assumptions

The element deformation is described using the same shape functions as in Ref. 51, hence the name isoparametric

$$\phi = \sum_i N_i'(\xi,\zeta,\eta)\phi_i \tag{168}$$

where ϕ_i represents the unknown generalized degrees of freedom associated with an individual element.

With the use of "relative coordinates" we now derive a single isoparametric hexahedron which has a variable number of nodes, of between 8 and 20. In practice this is especially useful where it is necessary to generate a mesh containing adjacent elements with different numbers of nodes, as occurs when a mesh changes character, e.g., going from a more coarse to a finer mesh. For a more detailed description of the use of relative coordinates see Ref. 52.

For an isoparametric hexahedron of between 8 and 20 nodes the shape function is as follows

$$\phi = \sum_i N_i' \phi_i \qquad i = 1, 2, \ldots, 8 \text{ plus any midside nodes} \tag{169}$$

where $N_i' = P_i$ for the midside nodes $(i = 9, \ldots, 20)$, as they exist, and $N_i' = P_i - \frac{1}{2}(P_I + P_J + P_K)$ for the corner nodes $(i = 1, \ldots, 8)$ where I, J, and K represent the midside nodes, as they exist, adjacent to corner node i.

The P functions are given as follows

$$P_i = \tfrac{1}{8}(1 + \xi_o)(1 + \zeta_o)(1 + \eta_o) \tag{170}$$

for the corner nodes and

$$\left.\begin{aligned} P_i &= \tfrac{1}{4}(1 - \xi^2)(1 + \zeta_o)(1 + \eta_o) & \xi_i &= 0 \\ P_i &= \tfrac{1}{4}(1 + \xi_o)(1 - \zeta^2)(1 + \eta_o) & \zeta_i &= 0 \\ P_i &= \tfrac{1}{4}(1 + \xi_o)(1 + \zeta_o)(1 - \eta^2) & \eta_i &= 0 \end{aligned}\right\} \tag{171}$$

for the midside nodes, where $\xi_o = \xi\xi_i$, $\zeta_o = \zeta\zeta_i$, and $\eta_o = \eta\eta_i$.

We can see from Eqs. (169) and (170) that the shape functions are built-up from an eight-node hexahedron representation (no midside nodes existing). When we have no midside nodes we arrive at

$$[B] = \begin{bmatrix} \frac{\partial N'}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N'}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N'}{\partial z} \\ \frac{\partial N'}{\partial y} & 0 & 0 \\ 0 & \frac{\partial N'}{\partial x} & 0 \\ \frac{\partial N'}{\partial z} & 0 & \frac{\partial N'}{\partial x} \end{bmatrix}$$

In order to evaluate the terms in the $[B]$ matrix we use the relationship

$$\begin{Bmatrix} \frac{\partial N'}{\partial x} \\ \frac{\partial N'}{\partial y} \\ \frac{\partial N'}{\partial z} \end{Bmatrix} = [J^{-1}] \begin{Bmatrix} \frac{\partial N'}{\partial \xi} \\ \frac{\partial N'}{\partial \zeta} \\ \frac{\partial N'}{\partial \eta} \end{Bmatrix} \tag{177}$$

where J is given by Eq. (175).

For an arbitrary element configuration the explicit integration represented by Eq. (174) cannot be carried out. We employ a Gauss quadrature numerical integration scheme as an efficient procedure (Ref. 45).

4.9.4 Geometric Stiffness Matrix

There is no geometric stiffness matrix currently available with this element in PLANS.

4.9.5 Initial Strain Stiffness Matrix

The initial strain stiffness matrix is given by

$$[k^*] = \int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} [B]^T[C](\det J)d\xi d\zeta d\eta \qquad (178)$$

where J is given by Eq. (175) and [C] is defined in Section 4.9.8 for this element. To obtain Eq. (178) the initial strains are assumed to be constant throughout the element and evaluated at the element centroid. The plastic load vector is given by

$$\{Q\} = [k^*]\{\epsilon_o\} \qquad (179)$$

4.9.6 Stresses and Strains

The element stresses are found at the centroid from

$$\{\sigma\} = [C]\{(\epsilon_T - \epsilon_o\}$$

They are given in the principal material directions. The total strains are obtained from Eq. (176). The initial strains are due to both plastic and thermal effects and are the same as used in deriving the effective load. The elastic coefficient matrix, [C], is given in Section 4.9.8 for this element.

4.9.7 Thermal Loads

Thermal loads can be applied to an element by specifying the nodal temperature, T, and coefficients of expansion, α_i. The thermal strains are $\epsilon_{xx} = \epsilon_{yy} = \epsilon_{zz} = \alpha_i T$, $\epsilon_{xy} = \epsilon_{yz} = \epsilon_{xz} = 0$. These are treated as initial strains and the load vector is obtained from Eq. (179).

The nodal temperatures are input only at corner nodes and are averaged to obtain an element temperature for use in the thermal load vector. The α_i may be different in each of the principal directions of orthotropy.

4.9.8 Material Properties

The elastic coefficient matrix, $[C]$, corresponds to an orthotropic material and can be expressed as

$$[C] = [Q]^T[C_o][Q] \tag{180}$$

where $[C_o]$ represents the reduced coefficient matrix along the principal axes of orthotropy and $[Q]$ represents the transformation into global coordinates. The terms of $[C_o]$ are as follows

$$C_{11} = E_1(1 - \nu_{32}\nu_{23})/\Delta$$

$$C_{12} = C_{21} = E_1(\nu_{12} + \nu_{32}\nu_{13})/\Delta$$

$$C_{13} = C_{31} = E_1(\nu_{13} + \nu_{12}\nu_{23})/\Delta$$

$$C_{22} = E_2(1 - \nu_{13}\nu_{31})/\Delta$$

$$C_{23} = C_{32} = E_2(\nu_{23} + \nu_{13}\nu_{21})/\Delta$$

$$C_{33} = E_3(1 - \nu_{12}\nu_{21})/\Delta$$

$$C_{44} = G_{12}$$

$$C_{55} = G_{23}$$

$$C_{66} = G_{13}$$

where

$$\frac{\nu_{12}}{E_2} = \frac{\nu_{21}}{E_1}$$

$$\frac{\nu_{13}}{E_3} = \frac{\nu_{31}}{E_1}$$

$$\frac{\nu_{23}}{E_3} = \frac{\nu_{32}}{E_2}$$

The transformation is taken first as a rotation about the z-axis, then about the x-axis, then about the y-axis, i.e.,

$$Q = Q_y Q_x Q_z$$

Three types of hardening laws may be specified for these elements. First, elastic-ideally plastic behavior based on Hill's criterion for orthotropic materials may be input. This requires inputing the three yield stresses in the principal directions of orthotropy and the three yield stresses in shear with respect to these planes. The two other types of hardening may currently be used for initially isotropic material behavior. These are linear strain hardening and nonlinear strain hardening based on a Ramberg-Osgood representation of the stress-strain law. At the end of each half-cycle of loading the plastic material properties may be changed.

4.9.9 Loadings

Two different types of loading conditions can be applied. A consistent load can be applied across a face of an element. A concentrated load can be placed at a node.

Surface Tractions — The consistent load vector is obtained from the last term of Eq. (173). For a consistent load we represent the applied tractions in the same functional form as the generalized unknowns, i.e.,

$$\{T\} = [N]\{T_i\} \tag{181}$$

where $\{T_i\}$ represents the applied nodal tractions. Substituting Eq. (181) into Eq. (173) and integrating in the local coordinate system we arrive at

$$\{F_i\} = \left[\int_{-1}^{1}\int_{-1}^{1} [N]^T[N]\Delta d\eta^1 d\eta^2\right]\{T_i\} \tag{182}$$

where η^1 and η^2 define the plane over which the traction is applied and

$$\Delta = \sqrt{A^2 + B^2 + C^2}$$

where

$$A = \left(\frac{\partial x^2}{\partial \eta_1}\frac{\partial x^3}{\partial \eta_2} - \frac{\partial x^3}{\partial \eta_1}\frac{\partial x^2}{\partial \eta_2}\right)$$

$$B = \left(\frac{\partial x^3}{\partial \eta_1}\frac{\partial x^1}{\partial \eta_2} - \frac{\partial x^1}{\partial \eta_1}\frac{\partial x^3}{\partial \eta_2}\right)$$

$$C = \left(\frac{\partial x^1}{\partial \eta_1}\frac{\partial x^2}{\partial \eta_2} - \frac{\partial x^2}{\partial \eta_1}\frac{\partial x^1}{\partial \eta_2}\right)$$

From Eq. (182) we see that the form of the load depends on whether midside nodes appear. For a face with no midside nodes the load varies linearly. When a midside node appears the load varies quadratically in the direction of the edge of the midside node. It is important to note that the input is the applied tractions T_i in Eq. (181), at nodes and not the total load.

Concentrated Loads - A concentrated load can be placed at any node by simply specifying its magnitude (total load) and direction. Thus, in this case we are actually inputting F_i.

4.9.10 General Comments

- Integration Order - A rectangular parallelepiped, in global coordinates, of eight nodes requires a 2 x 2 x 2 point Gauss integration while a rectangular parallelepiped of twenty nodes requires a 3 x 3 x 3 point Gauss integration. Care must, however, be taken as the element diverges from a parallelepiped. An exact stiffness can no longer be calculated but a good approximation can be found by increasing the order of integration. Orders of integration which are too low lead to elements that are too flexible. Orders of integration that are too high cause roundoff problems. It is suggested that when a choice has to be made the order of integration be kept on the low side. This eliminates any roundoff errors and since element stiffnesses are generally too high the error in the stiffness matrix is in the right direction. In general, it is best to keep the element as rectangular as possible.

- Orthotropic properties may only coincide with the global axis currently.

- This element is available in the HEX module.

19. Martin, H. C., "Finite Element Formulations of Geometrically Nonlinear Problems," in R. H. Gallagher, Y. Yamada, and J. T. Oden (Eds.), Proc. 1st U.S.-Japan Seminar on Matrix Methods in Structural Mechanics and Design (Tokyo, August 1969), UAH Press, Univ. of Alabama, Huntsville, Alabama, pp. 343-381, 1970.

20. Murray, D. and Wilson, E., "Finite Element Large Deflection Analysis of Plates," ASCE, J. Eng. Mech. Div., Vol. 95, No. EM1, pp. 143-165, February 1969.

21. Oden, J. T., "Finite Element Applications in Nonlinear Structural Analysis," Proc. ASCE Symposium on Application of Finite Element Methods in Civil Engineering (Nashville, November 1969), Vanderbilt Univ., Nashville, Tennessee, pp. 419-456, 1970.

22. Haisler, W., Stricklin, J., and Stebbins, F., "Development and Evaluation of Solution Procedures for Geometrically Nonlinear Structural Analysis," AIAA Journal, Vol. 10, No. 3, pp. 264-272, March 1972.

23. Marcal, P., "Large Deflection Analysis of Elastic-Plastic Shells of Revolution," AIAA Journal, Vol. 8, No. 9, p. 1627, 1970.

24. Hofmeister, L., Greenbaum, G., and Evensen, D., "Large Strain, Elasto-Plastic Finite Element Analysis," AIAA Journal, Vol. 9, No. 7, p. 1248, 1971.

25. Stricklin, J., Haisler, W., and Von Riesemann, W., "Evaluation of Solution Procedures for Material and/or Geometrical Nonlinear Structural Analysis by the Direct Stiffness Method," presented at the AIAA/ASME 13th Structures, Structural Dynamics, and Materials Conference, San Antonio, Texas, April 1972.

26. Levine, H. et al., "Nonlinear Behavior of Shells of Revolution under Cyclic Loading," J. Computers and Structures, Vol. 3, pp. 589-617, 1973.

27. Stricklin, J. A., Von Riesemann, W. A., Tillerson, J. A., and Haisler, W. E., "Static Geometric and Material Nonlinear Analysis," in J. T. Oden, R. W. Clough, and Y. Yamamoto (Eds.), Proc. 2nd U.S.-Japan Seminar on Advances in Computational Methods in Structural Mechanics and Design (Berkeley, August 1972), UAH Press, Univ. of Alabama, Huntsville, Alabama, pp. 301-324, 1972.

28. Armen, H., "Plastic Analysis," Proc. of Symposium on Structural Mechanics Programs - Surveys, Assessments, and Availability, edited by W. Pilkey et al., University Press of Virginia, June 1974.

29. Von Riesemann, W. A., Stricklin, J. A., and Haisler, W. E., "Nonlinear Continua," Proc. of Symposium on Structural Mechanics Programs - Surveys, Assessments, and Availability, edited by W. Pilkey et al., University Press of Virginia, June 1974.

30. Armen, H., Levine, H., Pifko, A., and Levy, A., "Nonlinear Analysis of Structures," NASA CR-2351, March 1974.

31. Armen, H. A. Jr., Saleme, E., Pifko, A., and Levine, H. S., "Nonlinear Crack Analysis with Finite Elements," in R. F. Hartung (Ed.), Numerical Solution of Nonlinear Structural Problems, AMD, Vol. 6, ASME Winter Annual Meeting, Detroit, November 1973.

32. Pifko, A., Levine, H. S., Armen, H., Jr., and Levy, A., "PLANS - A Finite Element Program for Nonlinear Analysis of Structures," Preprint No. 74-WA/PVP-6, Pressure Vessels and Piping Division, Transactions ASME, Winter Annual Meeting, New York, N.Y., November 17-22, 1974.

33. Hill, R., The Mathematical Theory of Plasticity, Chapter 12, Oxford University Press, 1950.

34. Prager, W., "The Theory of Plasticity: A Survey of Recent Achievements," (James Clayton Lecture) Proc. Instn. Mech. Engrs., Vol. 169, p. 41, 1955.

35. Prager, W., "A New Method of Analyzing Stress and Strains in Work-Hardening Plastic Solids," J. Appl. Mech., Vol. 23, p. 493, 1956.

36. Ziegler, H., "A Modification of Prager's Hardening Rule," Quart. Appl. Math., Vol. 17, No. 1, p. 55, 1959.

37. Ramberg, W. and Osgood, W. R., "Description of Stress-Strain Curves by Three Parameters," NACA TN-902, 1943.

38. Drucker, D., "Plasticity," Structural Mechanics, Proceedings First Symposium on Naval Structural Mechanics (Goodier, J. and Hoff, N., Eds.), Pergamon Press, p. 407, 1960.

39. Levine, H. S. and Svalbonas, V., "Inelastic, Nonlinear Analysis of Stiffened Shells of Revolution by Numerical Integration," Trans. of ASME, J. of Pressure Vessel Technology, Vol. 96, Series J, No. 2, May 1974.

40. Tsai, S. and Wu, E., "A General Theory of Strength for Anisotropic Materials," J. Composite Materials, Vol. 5, p. 58, January 1971.

41. Pugh, C. E., Liu, K. C., Corum, J. M., and Greenstreet, W. L., "Currently Recommended Constitutive Equations for Inelastic Design Analysis of FFTF Components, "Oak Ridge National Laboratory Report ORNL-TM-3602, September 1972.

42. Morrow, J., "Cyclic Plastic Strain Energy and Fatigue of Metals," ASTM STP No. 378, Amer. Soc. for Testing and Materials, 1965.

43. Sanders, J. L., Jr., "Nonlinear Theories for Thin Shells," Quarterly Applied Math., Vol. 21, No. 1, pp. 21-26, 1963.

44. Levine, H. S. and Armen, H., Jr., "A Refined Doubly Curved Axisymmetric Orthotropic Shell Element," Grumman Research Department Memorandum RM-496, February 1971.

45. Zienkiewicz, O. C., The Finite Element Method in Structural and Continuum Mechanics, McGraw-Hill Publishing Company, pp. 46-61, 1968.

46. Dwyer, W. J., "An Improved Automated Structural Optimization Program," AFFDL-TR-74-96, September 1974.

47. Garvey, S. J., "The Quadrilateral Shear Panel," Aircraft Engineering, May 1951.

48. Tocher, J. L. and Hartz, B. J., "Higher Order Finite Element for Plane Stress," J. Eng. Mech. Division, Proc. ASCE, Vol. 93, No. EM4, August 1967.

49. Bell, K., "Analysis of Thin Plates in Bending Using Triangular Finite Elements," Institutt for Statikk, Norges Tekniske Hogskole, Trondheim, Norway, February 1968.

50. Cowper, G. et al., "A High Precision Triangular Plate-Bending Element," Aeronautical Report LR-514, National Research Council of Canada, Ottowa, December 1968.

51. Zienkiewicz, O. C., "Isoparametric and Allied Numerically Integrated Elements - A Review," Numerical and Computer Methods in Structural Mechanics, Fenves, S. J., Perrone, N., Robinson, A. R., and Schnobrien, W. C., Eds., Academic Press, pp. 13-41, 1973.

52. Levy, A., A Three Dimensional 'Variable Node' Isoparametric Solid Element, Grumman Research Department Memorandum RM-587, July 1974.