

*NASA Conference Publication 3034
Part 1*

Computational Methods for Structural Mechanics and Dynamics

Edited by
W. Jefferson Stroud
Jerrold M. Housner
John A. Tanner
and Robert J. Hayduk
*NASA Langley Research Center
Hampton, Virginia*

Proceedings of a workshop sponsored by
the National Aeronautics and Space
Administration and held at
NASA Langley Research Center
Hampton, Virginia
June 19–21, 1985

NASA
National Aeronautics and
Space Administration
Office of Management
Scientific and Technical
Information Division

1989

PREFACE

This document contains the proceedings of the Workshop on Computational Methods for Structural Mechanics and Dynamics held at NASA Langley Research Center, June 19-21, 1985. The workshop was sponsored by NASA Langley Research Center.

The workshop had two objectives. The first objective was to introduce to the structural analysis technical community a new Langley research activity in structural analysis called Computational Structural Mechanics, or CSM. The second objective was to hear experts discuss important structural analysis problems and methods for solving those problems.

The workshop was organized into the following four sessions:

- | | | | | |
|-------------------------------------------|---|----------|---|---------|
| 1. Local/Global Nonlinear Stress Analysis | - | Full day | - | June 19 |
| 2. Tire Modeling | - | Half day | - | June 20 |
| 3. Transient Dynamics | - | Half day | - | June 20 |
| 4. Multi-Body Dynamics | - | Full day | - | June 21 |

Each session closed with a panel discussion.

Papers in these proceedings are grouped by session and identified in the contents. The order of the papers is the order of the presentations at the workshop. The proceedings also include any transcription of questions and answers that followed each paper and panel discussions that followed each session.

The use of trade names or names of manufacturers in this publication does not constitute an official endorsement of such products or manufacturers, either expressed or implied, by the National Aeronautics and Space Administration.

W. Jefferson Stroud
Jerrold M. Housner
John A. Tanner
Robert J. Hayduk
Workshop Co-Chairmen

PRECEDING PAGE BLANK NOT FILMED

CONTENTS

PREFACE	iii
CONTENTS	v
INTRODUCTION: Computational Structural Mechanics Activity	1

PART 1

LOCAL/GLOBAL NONLINEAR STRESS ANALYSIS

SOME ISSUES IN NUMERICAL SIMULATION OF NONLINEAR STRUCTURAL RESPONSE	7
H. D. Hibbitt	
COMPUTERIZED STRUCTURAL MECHANICS FOR 1990'S: ADVANCED AIRCRAFT NEEDS	25
A. V. Viswanathan and B. F. Backman	
NONLINEAR RESPONSE OF A BLADE-STIFFENED GRAPHITE-EPOXY PANEL WITH A DISCONTINUOUS STIFFENER: WORK IN PROGRESS	51
Norman F. Knight, Jr., William H. Greene, and W. Jefferson Stroud	
COMPUTATIONAL PROCEDURES FOR POSTBUCKLING OF COMPOSITE SHELLS	67
G. M. Stanley and C. A. Felippa	
A REVIEW OF SOME PROBLEMS IN GLOBAL-LOCAL STRESS ANALYSIS	89
Richard B. Nelson	
SOME COMMENTS ON GLOBAL LOCAL ANALYSES	103
Satya N. Atluri	
ON COMPUTATIONAL SCHEMES FOR GLOBAL-LOCAL STRESS ANALYSIS	123
J. N. Reddy	
GLOBAL FUNCTIONS IN GLOBAL-LOCAL FINITE-ELEMENT ANALYSIS OF LOCALIZED STRESSES IN PRISMATIC STRUCTURES	135
Stanley B. Dong	
GLOBAL-LOCAL METHODOLOGIES AND THEIR APPLICATION TO NONLINEAR ANALYSIS	151
Ahmed K. Noor	
APPLICATION OF THE P-VERSION OF THE FINITE-ELEMENT METHOD TO GLOBAL-LOCAL PROBLEMS.....	169
Barna A. Szabo'	
QUESTION AND ANSWER SESSION	179
PANEL DISCUSSION	190

PRECEDING PAGE BLANK NOT FILMED

TIRE MODELING

FEATURES AND CHARACTERIZATION NEEDS OF RUBBER COMPOSITE STRUCTURES	211
Farhad Tabaddor	
NONLINEAR HIERARCHICAL SUBSTRUCTURAL PARALLELISM AND COMPUTER ARCHITECTURE	249
Joe Padovan	
ADAPTIVE METHODS, ROLLING CONTACT, AND NONCLASSICAL FRICTION LAWS	269
J. T. Oden	
CONTACT SOLUTION ALGORITHMS	291
John T. Tielking	
EXPLOITING SYMMETRIES IN THE MODELING AND ANALYSIS OF TIRES	317
Ahmed K. Noor, C. M. Andersen and John A. Tanner	
PANEL DISCUSSION	331

PART 2*

TRANSIENT DYNAMICS

SUMMARY: TRANSIENT DYNAMICS	341
Robert J. Hayduk	
IMPROVING TRANSIENT ANALYSIS TECHNOLOGY FOR AIRCRAFT STRUCTURES	343
R. J. Melosh and Mladen Chargin	
EXPLICIT, IMPLICIT, AND HYBRID METHODS	355
T. Belytschko	
UNCONDITIONALLY STABLE CONCURRENT PROCEDURES FOR TRANSIENT FINITE-ELEMENT ANALYSIS	369
Michael Ortiz and Bahram Nour-Omid	
TRANSIENT ANALYSIS TECHNIQUES IN PERFORMING IMPACT AND CRASH DYNAMIC STUDIES	383
A. B. Pifko and R. Winter	
APPLICATION AND IMPLEMENTATION OF TRANSIENT ALGORITHMS IN COMPUTER PROGRAMS	407
David J. Benson	
QUESTION AND ANSWER SESSION	419
PANEL DISCUSSION	425

*Presented under separate cover.

MULTI-BODY DYNAMICS

RECENT DEVELOPMENTS IN DEPLOYMENT ANALYSIS SIMULATION USING A MULTI-BODY COMPUTER CODE	435
Jerrold M. Housner	
DYNAMIC ANALYSIS OF SPACE STRUCTURES INCLUDING ELASTIC, MULTIBODY, AND CONTROL BEHAVIOR	443
Larry Pinson and Keto Soosaar	
Part 1: DYNAMICS OF FLEXIBLE MULTI-BODY MECHANISMS AND MANIPULATORS: AN OVERVIEW	455
Steven Dubowsky	
Part 2: APPLICATION OF FINITE-ELEMENT METHODS TO DYNAMIC ANALYSIS OF FLEXIBLE SPATIAL AND CO-PLANAR LINKAGE SYSTEMS	459
Steven Dubowsky	
DYNAMICS OF ARTICULATED STRUCTURES	491
Edward J. Haug	
MODAL REDUCTION STRATEGIES FOR INTERCONNECTED FLEXIBLE BODIES SIMULATION	517
F. O. Eke and G. K. Man	
COMPUTATIONAL ASPECTS OF MULTI-BODY DYNAMICS	527
K. C. Park	
CONSTRAINT ELIMINATION IN DYNAMICAL SYSTEMS	537
R. P. Singh and P. W. Likins	
NONLINEAR CHARACTERISTICS OF JOINTS AS ELEMENTS OF MULTI-BODY DYNAMIC SYSTEMS	543
Edward F. Crawley	
QUESTION AND ANSWER SESSION	571
PANEL DISCUSSION	589

PART 1

INTRODUCTION

Computational Structural Mechanics Activity

NASA Langley has initiated a new research activity in structural analysis called Computational Structural Mechanics (CSM). The broad objective of the CSM activity is to develop advanced structural analysis technology that will exploit modern and emerging computers, such as computers with vector and parallel processing capabilities. There are three main research activities under way in CSM: (1) parallel processing, (2) software test bed, and (3) structural analysis methods. The following three paragraphs provide a brief description of the three main CSM research activities.

Parallel processing.- Within the next 5 years, all new high-performance computers will have multiple processors. CSM researchers are developing software approaches and structural analysis algorithms that will exploit this new capability. This research activity has been under way for several years and has used an experimental, in-house-developed multiprocessor computer known as the finite-element machine (FEM). At this writing, FEM has 16 processors. A new commercial multiprocessor computer will be delivered to Langley for parallel processing research. The parallel processing research activity involves several in-house researchers and universities.

Software test bed.- The test bed is to be a modern, modular system that handles data efficiently, that contains a command language which is powerful and easy to learn and use, and that has an architecture which allows users to add software with minimal difficulty. The test bed will be used to study the ingredients of modern software and how those ingredients should fit together. More importantly, the test bed will be used to study and evaluate structural analysis methods on practical applications problems. The test bed research is just now getting under way. It involves in-house researchers and contractors.

Structural analysis methods.- The structural analysis methods research has several goals. One goal is to develop analysis methods that are general. This goal of generality leads naturally to finite-element methods, but the research will also include other structural analysis methods. Another goal is that the methods be amenable to error analysis; that is, given a physical problem and a mathematical model of that problem, an analyst would like to know the probable error in predicting a given response quantity. The ultimate objective is to specify the error tolerances and to use automated logic to adjust the mathematical model or solution strategy to obtain that accuracy. A third goal is to develop structural analysis methods that can exploit parallel processing computers. Our structural analysis methods research will focus initially on three types of problems: local/global nonlinear stress analysis, nonlinear transient dynamics, and tire modeling. These three types of problems are the topics of the workshop and are discussed in detail subsequently.

Workshop Description

This workshop had several objectives, among which were

1. To introduce CSM to the structural analysis technical community, particularly the university community

2. To hear experts discuss important structural analysis problems and methods for solving those problems
3. To help make decisions regarding research thrusts in structural analysis methods development

Three types of problems were addressed at the workshop: local/global nonlinear stress analysis, nonlinear transient dynamics, and tire modeling. The following is a description of these three types of problems.

Local/global nonlinear stress analysis.- A local/global stress analysis is a local, detailed stress analysis within a larger, less-refined analysis model. Both the application and the solution procedure are taken to be arbitrary. The definition of local/global is not precise. Because of this ambiguity, two focus problems are used to define (for the workshop) what is meant by the words local and global. The first focus problem is a stiffened flat composite panel with a circular hole that causes a stiffener to be discontinuous. The panel is subjected to an in-plane compressive load in the direction of the stiffeners. The second focus problem is similar, except that it is curved and unstiffened. In each case, the overall response of the panel is the global problem; the response near the hole is the local problem. In their talks, speakers might not have presented solutions to the workshop focus problems, but their comments regarding analysis methods referred to the focus problems.

Structural problems requiring a local/global stress analysis generally involve discontinuities which cause rapid changes in stress. The high stress gradients are a local phenomenon. Analysis procedures that must be used to predict these stress gradients are not required away from the discontinuity. In regions in which stresses vary slowly, less-refined analyses are adequate. Practical considerations demand a multilevel approach. It is these various levels of analysis that cause difficulties in the local/global problem. In the vicinity of the discontinuity, an analyst may refine a finite-element grid, introduce a three-dimensional grid, use more powerful elements, apply a classical solution, and use other analysis tools and approaches. Usually, the analyst will take additional steps to insure that his analysis is adequate. He may rework the problem with a modified model, or he may make comparisons with the results from other analysis procedures.

To help the structural analyst overcome the difficulties just described, a systematic analysis procedure is needed that is amenable to error analysis. The analyst would like to prescribe the error tolerances in predicting the stress at specified locations and use automated logic to adjust the mathematical model or solution strategy to obtain that accuracy. In addition, an analyst needs the capability to calculate derivatives of response quantities with respect to parameters that define the problem. Such a sensitivity analysis is necessary for design purposes and is helpful in determining the validity of an analysis. Finally, this systematic analysis procedure that includes error analysis, adaptive solution refinement, and sensitivity analysis should be easy to use. The goal is for solutions to local/global problems, as well as other structural analysis problems, to become routine.

Tire modeling.- One of the most challenging problems in the field of Computational Structural Mechanics is predicting the response of automotive and aircraft tires during ground handling operations. The pneumatic tire exhibits a complex geometry which can generally be described as a noncircular, incomplete torus. Under normal operational conditions, these tires are subjected to large deformations and

structural rotations. The heavy weight loading requirements imposed on many aircraft and off-road vehicle tires require their carcasses to be so thick that a significant portion of the resulting strain energy is attributed to transverse shear deformation. Furthermore, the most interesting loading conditions associated with braking and steering operations are nonsymmetric.

The tire is a laminated composite structure consisting of rubber and textile constituents. The material properties of the constituents may differ by as much as 2 or 3 orders of magnitude, thereby pushing the limits of current composite theory. The rubber exhibits nearly incompressible characteristics and is subject to large hysteresis effects. Under dynamic loading conditions, these hysteresis effects can lead to large thermal gradients in the tire carcass and, hence, significantly affect the material properties of the constituents. Furthermore, the material properties of the tire carcass can be highly anisotropic and nonhomogeneous.

A central feature of any tire model must be a sophisticated contact algorithm which is capable of handling contact friction forces. The tire designer will also require the model to predict stress distributions in critical regions of the tire carcass. This requirement might necessitate the local integration of three-dimensional finite elements into an otherwise two-dimensional model. Finally, the model must demonstrate its cost effectiveness before it is generally accepted by the United States tire industry.

Nonlinear transient dynamics.- During the last 2 decades there has been a growing interest in the numerical solution of time varying equations of structural and mechanical systems. Such interest comes from such diverse fields of application as impact mechanisms, robotic manipulation, aircraft structures and space structures. These applications often involve the response of systems composed of many structural or mechanical components.

Dynamic analyses of these systems involve both the formulation of the governing equations and their solution. Usually, the governing equations are discretized spatially and then integrated temporally using one of a wide variety of time integration procedures. It is therefore reasonable to consider the dynamic analysis as consisting of two activities: a formulation and spatial modeling activity and a temporal solution activity.

The formulation and spatial modeling activity involves treatment of constraints, controls, possible large deformations of components and large relative angular motion between components. It is not unusual that the formulation and modeling results in large sets of nonlinear governing equations. Consequently, the temporal solution of such systems can be computationally intensive, and it becomes the aim of the formulation and modeling activity to improve computational efficiency and computer implementation.

The solution of the governing equations involves the temporal integration of equations of motion subject to initial conditions and constraints. The integration is usually carried out using either some type of modal superposition procedure or a direct time integration procedure. The modal superposition procedure, though very powerful for the solution of linear equations, loses much of its effectiveness in dealing with nonlinear equations and for this reason is not included in this volume. Generally, direct time integration procedures are used for dynamic analysis of mechanical and structural systems governed by nonlinear equations.

Direct time integration procedures are usually referred to as algorithms since it is the computation of the response quantities which are sought by these procedures rather than theorems relating to solution existence, uniqueness, etc. A very large number of these algorithms exist. Their popularity is attested to by the inclusion of such algorithms in nearly all general-purpose computer programs for analyzing structures, mechanisms, and satellites. Many such programs offer the user a selection of algorithms. However, for the solution of problems involving the temporal integration of a large number of equations, the use of some of these algorithms leads to very intensive computer usage due to a lack of computational efficiency.

Not only is solution efficiency in need of improvement, but so is solution reliability. The user desires a certain level of confidence that the computed results are reasonably accurate. Unlike problems of statics, solution satisfaction of the governing equations does not provide a guarantee that the solution is correct, for the solution must come from a prescribed set of initial conditions. Conservation of energy affords a level of confidence in the solution, if such is applicable, but in many problems it is not applicable. Thus, both efficiency and reliability are items which need improvement.

Research into developing improved formulations and modeling and their solution is presently being performed. However, much remains to be done. Furthermore, it takes considerable time for such developments to find their way into computer programs which are used in routine practice. Yet the application needs to continue to grow.

Two sessions in this research area were held. One half-day session dealt exclusively with solution procedures, while one full-day session dealt with development and treatment of items critical to proper formulation and modeling of multi-body systems.

Thirty papers dealing with the three classes of problems just discussed were presented at the workshop. In these papers, solution concepts are reviewed, computational algorithms are outlined, an assessment of various methodologies is made, and problem areas are identified. In addition, directions for future research are recommended.

LOCAL/GLOBAL NONLINEAR STRESS ANALYSIS

Some Issues in Numerical Simulation of Nonlinear Structural Response

H. D. Hibbitt .
Hibbitt, Karlsson and Sorensen
Providence, R.I.

1. Introduction

For some considerable time the author has been involved in the development of commercial finite element software, and this paper is written from that perspective. One of the remarkable features of the finite element method is its generality, and there is no better reflection of this than the observation that our company and several of our competitors each market one product only—a single, “general purpose” finite element code (our particular code is called Abaqus). These codes provide practical tools that are used in an astonishingly wide range of engineering applications that include critical aspects of the safety evaluation of nuclear power plants or of heavily loaded offshore structures in the hostile environments of the North Sea or the Arctic, major design activities associated with the development of airframes for high strength and minimum weight, thermal analysis of electronic components, and the design of sports equipment. For various reasons, the code that my company develops and markets is generally used in the area of more advanced applications. These applications almost always involve nonlinear effects. There is little doubt in my mind that the need for such analysis will continue to grow—it is very easy to identify problems which should be reachable in the next generation or two of computers and software and which have substantial economic importance.

The development, maintenance and support of production software involves many activities, but—at least in the more advanced application areas where we try to contribute—the effectiveness of our product depends critically on the quality of the mechanics and mechanics related algorithms that we implement. It is generally true that the end users are not sophisticated with respect to what is now being called “computational mechanics.” They have other interests and motivations, not the least of which is the need to complete work successfully and to a schedule. Thus, “algorithmic robustness” is of primary concern to us: we should choose those methods that we believe will maximize reliability with minimal understanding on the part of the user. Computational efficiency is important because there are always limited resources, and hence problems that we would like to do but which are too time consuming or costly. In reality, we compromise: for example, we

PRECEDING PAGE BLANK NOT FILMED

knowingly commit what Strang and Fix (1973) call "variational crimes" because we get away with them often enough for it to be worthwhile. But robustness implies a need for thorough understanding of the algorithms: we should at least know where the limitations of an algorithm are likely to be. This is not a simple task: for example, we still do not have practical ways of assessing local error, even in a linear numerical solution.

It is easy to identify important practical problems that, presently, are computationally rather difficult, but which will become relatively routine in the not too distant future. Two that are taking up much of our time just now are the problem of simulating vehicle (especially automobile) crashes, and the simulation of rather complicated contact situations, such as the analysis of threaded connectors in drill pipe or casing which is subjected to very large axial forces, causing possible thread jump and quite substantial strains in the pipe. Both problems are modeled today on current generation computers (the Cray-1 and X-MP) with Abaqus and other codes. They challenge the limits of the algorithms in our code, and are computationally intensive—typical run times are several hours per case. The observations made in this paper are based on our experience to date with such problems.

Large scale general purpose codes have a rather long life span: two codes that are widely used at this time (Ansys and Nastran) were begun in the mid 1960's. Thus, in designing such a code, it is important to try to anticipate what sort of computers will be in general use for such applications in 10–15 years. Based on past history, this is a difficult extrapolation—computers are still developing at a rapid rate. We can make some guesses. For example, at Cray Research's Science and Engineering Symposium held in Minneapolis in 1985 Seymore Cray discussed the specification of the Cray-3, which he expects will be available in three years. The most relevant parts of that specification from our point of view are the size of the high-speed, directly addressable, memory (10^9 words), and the machine's parallelism. Consider a problem of order 50000 unknowns: many important cases of the types mentioned above are of this size or smaller. The rms half-bandwidth in such a case would be about 5000, so that the assembled symmetric part of the Jacobian (stiffness) matrix will occupy 250×10^6 words—a quarter of the available memory; the complete matrix will occupy half the memory. (More and more problems are likely to need the full Jacobian matrix in the context of Newton methods, because of the use of more realistic constitutive models, such as damage mechanics models for brittle or composite materials, which have a non-symmetric Jacobian). Element matrices, state variables for constitutive calculations, etc. are unlikely to occupy more than 5×10^6 words in such a case. Thus, it would seem that, for such applications, we need not worry about "I/O" problems—we can assume the model can run "in-core." This means that, for our purposes, the practical computational limitation will be the time taken to do the arithmetic. Cray Research and others seem to be moving rapidly toward parallelism to provide arithmetic speed. This should fit well with that part of our finite element codes that perform element and constitutive calculations: there we process a large number of such calculation points (perhaps 4000 elements, 30000 integration points in the 50000 degree-of-freedom case) so that, provided the code is designed to allow inner loops to spread over the processors, load balancing should not be much of a problem. It is interesting to note that, in this part of the code, typical vectors are of order 10–100, so that any vector processor with a long start-up time (like the Cyber 205) is not very suitable. The solution of the linear equations

is not such a natural fit into parallel architecture, but the problem is such an important one that it is likely that very effective modules will become available (such as Floating Point Systems now provide for their processors). Since we in fact are solving a nonlinear system, it may be that non-Newton methods, not requiring the resolution of a Jacobian matrix, may be preferable in such an environment. I assume that load balancing is the critical issue in a multi-processor machine, and that the most effective approach to that issue is at the inner loop level, rather than the macro (finite element) level.

The remainder of the paper discusses three principal areas. First, the paper discusses our experiences with the approaches we use to the two initial value problems of primary interest here—static and dynamic nonlinear response of structures. It then has a brief section on shell modeling. Then it discusses our current approach to the constitutive integration problem, in the context of conventional plasticity models. Finally, the paper lists some areas where we hope that research work will provide us with new methods, or with improvements to our present approaches.

2. Experience with initial value problems

2.1 Structural dynamics

The nonlinear dynamic problems that we most often see involve globally nonlinear response—most of the structure yields and undergoes finite rotations and, possibly, finite strains. However, locally nonlinear dynamic problems are not uncommon: in fact they represent a sufficiently important application area that we have found it worthwhile to provide methods to model them with some efficiency. A good example of such a case is the flow induced vibration of a steam generator tube in a nuclear plant. The tube rattles in its support plates due to vibration excited by instabilities in the coolant flow, and this intermittent contact causes wear and, eventually, may lead to leakage. The tube itself may be “sprung”—that is, the initial stress in the tube may be large enough that it forms a significant contribution to the tube’s stiffness—but the vibration amplitude is not large enough that the stiffness of the tube changes significantly during the motion. The nonlinearities are therefore confined to the support interaction, where they involve intermittent, “chattering” contact and friction. Our approach to such problems has been through conventional Guyan reduction, retaining the transverse displacements at the support points and enough other degrees of freedom in the tube to model its response accurately in the frequency range of interest (up to about 150 Hz in this case). The Guyan method has two drawbacks—it is based on guesswork, and the reduced model may be much bigger than necessary. Since the majority of the applications that have come to our attention involve relatively small models (typically less than 1000 degrees of freedom, with 30–80 degrees of freedom directly involved in local nonlinearities), these limitations are not very critical: it is not much effort to extract the modes of interest on the full model and then to experiment with reduced models until a reasonably small one is found that provides adequate matching.

Nonlinear dynamic problems are integrated explicitly or implicitly. The explicit approach brings with it several advantages because it is always used with a lumped mass

matrix and so no equations need be solved: this leads to great simplification in the computer code. But conditional stability is a serious limitation, especially in structural models where the thickness of the shell is usually the determining factor with respect to the time step. Many of the problems that we see are “event and response” cases, in which an initial input of energy is mostly dissipated as inelastic response in the structure. In these cases the response usually damps out fairly quickly into a pattern of plastic hinges, until some later disturbance—perhaps a secondary impact, or a sudden effect associated with geometry changes—causes this pattern to undergo a redistribution. Indeed, this phenomenon of plastic hinge formation is so much a part of such problems that it has given rise to the successful mode form solution approach of P. S. Symonds, J. B. Martin, and others (Symonds, 1967). It is difficult to accept that a conditionally stable integration method which can never expand the time step beyond some fraction of the shortest period exhibited by the model provides the optimal approach for such an application. My own experience with explicit methods is limited, so that I do not know how one deals with constraints (such as arise in mixed element formulations) within these methods. But if the methods are only useful when we do not solve equations, it seems that the generality of their application is limited.

Implicit methods are chosen for their numerical stability. Stability is usually discussed in the context of linear systems, and I am not sure of what stability proofs exist for the nonlinear problems we are trying to model. My own practical experience has been that numerical stability of the operator has never been a limitation: except for accuracy considerations, the time step is most usually limited by our ability to solve the equations. I will return to the equation solving problem later, because it is, in our experience, a key issue. Implicit methods have the great advantage of generality—the time step can be chosen for modeling reasons. And since we have to solve equations anyway, we are at liberty to introduce whatever additional equations we care to, including, for example, Lagrange multipliers to impose constraints. In Abaqus we have for some time been using the Hilber-Hughes-Taylor (1978) operator (an extension of the trapezoidal rule that provides the ability to introduce some numerical damping) with a simple “automatic” time step selection scheme, and this approach has been of great practical benefit. Again, consider the car crashworthiness problem. A front end collision test case for a car design usually involves about 10–15 milliseconds of response. With the shell elements in Abaqus and a mesh that is adequate to model the response usefully, the stability limit of the central difference operator is in the microsecond range. Using the implicit method, the analysis is usually completed in 150–300 time increments, which range from a few microseconds just after a major impact up to a significant fraction of a millisecond during the fixed plastic hinge régime. The utility of the method then depends on whether we can solve the nonlinear equations 150–300 times in an acceptable amount of computer time.

2.2 Statics

Smoothly nonlinear static problems are not uncommon, but it seems that many important applications involve abrupt changes in the response, which is often unstable as well. The car crash problem again serves to provide an example. During a front end crash, most conventional designs of front rails in cars are responding unstably during about 80% of

their usable deformation, and the switch into this collapsing response occurs quite sharply, as a buckling of the structure (which is a shell), after it has undergone considerable plastic deformation. Most shell buckling cases involve relatively thin shells, so that the equilibrium equations are not well conditioned. Elastic-plastic buckling often involves sudden “localization” of the plastic deformation into narrow regions, while major parts of the structure, which were yielding in the pre-collapse phase, unload elastically. Such problems make demands on the solution strategy—it must be able to detect, and to switch into, the alternate equilibrium path, and the instability of the collapse phase of the response must be handled. For the latter purpose we have found that our version of a “Riks” algorithm (Riks, 1979) has been most valuable. We use it with an “automatic” incrementation scheme, and find that it can often march right through into the collapse as far as we want to go, without too much stuttering. The most common complaint that we receive is that the solution sometimes turns back on itself. When this occurs it is at critical points where, presumably, the equilibrium path has very high curvature. For the present we ascribe this to a weakness in our implementation only: the method finds an equilibrium path, but it is not the path of interest. We have nothing in our code that can detect the possibility of alternative equilibrium paths: for example, we know we cannot obtain sensible results for a round bar in compression (Euler buckling) or tension (necking), without knowing that the switch may occur and seeding the problem definition with a suitable imperfection. A robust algorithm would not need this.

Rate form constitutive models (such as conventional plasticity or visco-plasticity theories) still must be integrated, even though the overall problem is quasi-static. There seems to be relatively little motivation for choosing explicit methods in general for this aspect of the problem, although there are some particular cases where an explicit approach makes sense: for example, many high temperature creep problems associated with metal structures can be treated efficiently with explicit integration of the creep model, because in these cases the response times of interest are usually not very long compared to characteristic relaxation times for the material subjected to the stresses that arise in the structure—otherwise the design would be unacceptable anyway. In Abaqus, except for this particular case, we use implicit integration for rate plasticity models, as discussed below.

3. Equation solving

Almost all of the procedures we use in Abaqus are based on a “full” Newton method. We have tried a few alternatives (modified Newton and quasi-Newton methods), but so many problems of interest are not very well conditioned and exhibit such knotty response that we have not looked at many of our standard test cases before we have rejected the alternatives that we have tried, and returned to the quadratic convergence of the full Newton method. However, our work is hardly rigorous or complete.

A significant part of the usage of our code involves systems for which the Jacobian matrix of the Newton method is not symmetric. Examples are the non-associated flow plasticity models that are often used in soil mechanics applications, and loading cases with “follower forces”, like Morison drag on offshore pipes and risers. In these cases we usually form and solve the non-symmetric system. There are some problems where we offer the

possibility of approximating the Jacobian with its symmetric part because our experience has suggested that many typical cases of that class are analysed with less computational expense in that way. An example is mentioned later in this paper in connection with integration of plasticity models. But we always provide the possibility of invoking the non-symmetric capability if needed, typically by restarting the simulation at some point, because it often happens that the symmetric approximation becomes less effective as the solution develops.

Newton's method introduces difficulties of its own. The most expensive is the need to form and solve a system of linear equations at each iteration, and the most awkward is the need to define the Jacobian matrix. The algebraic manipulations involved in defining this matrix can be formidable—and there are obviously cases when it cannot be defined, except perhaps numerically. For example, a recent paper on numerical methods in plasticity (Simo and Ortiz, 1984) contains the remark:

“In general, however, the task of evaluating the consistent tangent moduli in closed form may prove exceedingly laborious. It would appear, therefore, that a general purpose implementation of the physically more compelling algorithm . . . may require the use of quasi-Newton or secant-Newton methods . . .”

This difficulty should not be underestimated.

The straightforward approach we have been using for conventional plasticity models is discussed in Section 5. **One further comment concerning that method is appropriate here:** even in cases when the rate plasticity model exhibits the usual symmetry property obtained from assuming associated flow and a smooth yield surface, the exact Jacobian of the integrated model—using the integration operator that we have chosen—is not always symmetric.

The full Newton method is expensive computationally. We can provide some numbers to quantify this on computers that are available today. Abaqus is a general purpose code and is used on many different computer systems, presently ranging from the Apollo 300 to the Cray X-MP. The code is not particularly “tuned” to any system, and therefore should be representative of typical straightforward finite element codes (we know from benchmarks that this is generally true). We use some standard benchmark problems to estimate performance of our code on various computers: experience has shown that these benchmarks are reliable. One of these is a shell model, with 1000 eight-node elements. It has about 17000 degrees of freedom and an rms wavefront of 400 degrees of freedom. Such a mesh would be adequate for the typical front-end collision analysis that I used as an example above. On a CrayX-MP this problem takes 48 cp seconds per iteration: 60% of that time is in the linear equation solver, and most of the rest is associated with element and constitutive calculations. A typical dynamic analysis of a front end collision needs 150–300 time increments with the implicit operator that we use, and between three and four iterations per increment with the “almost full Newton” implementation of the shell in Abaqus (the initial stress terms associated with bending are not defined exactly in the Jacobian—there is no basic problem, we have simply not completed some lengthy manipulations. I do not think the terms we omit are very significant). Thus, such a crash

simulation can be expected to pass through the basic loop about 500–1200 times if our algorithms for time stepping, impact, etc. all are working well. This implies about $6\frac{1}{2}$ to 16 hours on the Cray for the job.

4. Shells

A substantial part of the modeling for which Abaqus is used involves shells, much of it in cases where geometric and material nonlinearities dominate the response. We have tried to provide useful capabilities for shell modeling, but we know that there are serious deficiencies in what we offer. Irons wrote often about shells: his definitive account of SemiLoof (Irons, 1976) and his textbook (Irons and Ahmad, 1980) both contain succinct statements of the difficulties. My impression is that things are improving rapidly. I hope so: I think we are all aware of the need.

In Abaqus we always treat shells as shells—we do not have any numerically degenerated solid elements acting as shells. The elements are formulated in terms of forces and moments at integration points. The behavior of the cross section is defined in closed form, or by numerical integration through the thickness. In most cases we use shear deformable elements, but the transverse shears are not considered in the constitutive calculations—we assume the shell is thin enough that transverse shear is not very important: it is simply a numerical technique that allows us to manage with low order interpolation. Low order elements are often desirable in practical cases.

Abaqus includes three types of shell model: axi-symmetric shells with axi-symmetric deformations, general shells, and pipes and elbows with deforming sections (ovalization and warping). These last elements are a rather special case which turns out to be important in certain piping applications that arise in the nuclear power industry and in the offshore oil industry. We often see problems where a mixture of shell and solid modeling is needed (K-joints in offshore platforms; Tees in pressure vessels). We have a standard technique for joining the shells to the solids, based on kinematic assumptions introduced as constraints. The approach appears to be satisfactory for the small strain cases where we have seen results.

The purely axi-symmetric case is rather simple because the problem is one dimensional. For this case we use a linear interpolation element with one integration point and a quadratic interpolation element with two. Our impression is that these elements are quite effective. Our implementation of these elements is based on a simplified finite membrane strain theory. The theory is similar in concept to Rodal's thesis (Rodal and Witmer, 1979), except that we use somewhat different strain measures because our applications involve material models for which logarithmic strain seems to be appropriate, and, in a case like this where the principal directions do not rotate, it is easy to work directly in terms of this strain. The main simplifying assumptions in the theory are that the thinning of the shell is uniform through the thickness and is defined by an incompressibility assumption on the reference surface of the shell, and that the thinning occurs smoothly, so that we can neglect gradients of thickness change with respect to position.

There are many applications for which a finite membrane strain model should be useful—obvious examples arise in sheet metal forming processes. The axi-symmetric case

is particularly simple and our version of a finite membrane strain theory is quite basic. Nevertheless, it takes two pages of the Abaqus theory manual just to write out the definition of the initial stress matrix, and the manipulations involved in reaching that definition are lengthy. The extension of the same formulation to general shell deformations will involve substantial algebraic manipulation.

The general shell elements that we use are the Batoz triangle, and the four and eight node "Ahmad quadrilaterals" (shear flexible elements using reduced integration). Our implementation of the Batoz triangle uses Batoz' interpolation functions in a co-rotational frame for bending, together with constant membrane strain. The element is in Abaqus because it is essential for us to offer a three-node triangular element for shell problems, and we have been advised that this element is among the better elements of this category. The element appears to behave well in bending—it converges rapidly and is relatively insensitive to distortion. The shortcomings of our implementation of the element are the constant membrane strain assumption, the faceted geometric approximation, and the need to use three integration points, this last because it makes the element more expensive than it should be, given the constant membrane strain assumption.

The Ahmad quadrilaterals are very attractive for cases involving material nonlinearity because the use of reduced integration minimizes the constitutive evaluations needed to form the element. The basis functions are also very simple. The four node element is of limited value without hourglass control: we use the hourglass controls proposed by Belytschko (Belytschko and Tsay, 1983) for this element. This improves the situation, but it is still not entirely satisfactory—we still find it necessary to change the hourglass stiffnesses from time to time, without being too sure of the reasons for the values we choose. Abaqus includes a print option which summarizes the energy content of the solution. It is not uncommon to see relatively significant "artificial strain energy" associated with the hourglass control: in typical high energy dynamic events this artificial strain energy may even exceed the residual "physical" energy (strain energy plus kinetic energy) at the end of the event. Yet the overall predictions of the significant aspects of the response are generally usable.

As Irons pointed out in his last paper (Irons and Loikkanen, 1983), the eight node Ahmad element does not pass the patch test except as a rectangle. We offer this element for historical reasons—it seemed to be a good choice at the time. In fact it does respectably well on shells that are not too thin, even if it is somewhat distorted. But there is another practical objection to the eight node interpolation scheme: contact problems are awkward with such functions. Contact over the entire element does not present serious difficulties, but, with the contact algorithms that we use (which have Lagrange multipliers to represent the contact pressure), we do not know how to deal with partial contact over an element—that is, with the possibility that contact or separation may occur over part of the surface represented by an element. We plan to add the nine-node version of the same element to overcome this last objection, even though Irons has told us that this form of the element only passes the patch test when it is geometrically bilinear (which would soon not be the case in a large displacement analysis, even if it were true to start with).

Both of the Ahmad elements, as we have implemented them, perform poorly if they are distorted and the shell is thin. This is a serious objection, but it would appear that

the element of K. C. Park (Park and Stanley, 1984) alleviates this problem. I think it is important, for practical cases, that we retain the simplicity of the low order elements as well as the cost effectiveness of reduced integration, especially when relatively severe nonlinearities (requiring, for example, modeling of finite strain effects) are present.

5. Integration of Conventional Plasticity Models

Conventional plasticity models present an interesting integration problem: the material behavior changes drastically at yield; the plastic strain is defined only as a rate, and the stress measures usually used are defined on the current configuration, so that the rotation of the material must be accounted for in some way. The problem has been receiving attention recently (for example Ortiz and Popov, 1984), and new approaches have been suggested. I would like to describe the approach that we have been using in Abaqus: overall, we are reasonably satisfied with it, but it raises some questions that we have not been able to answer, and which may also be relevant to the more recently proposed approaches.

Abaqus is primarily an implicit code, so that it is desirable to use an algorithm that is unconditionally stable. This is not strictly essential. In typical cases involving metals under ordinary conditions, our experience has been that we are unlikely to succeed in using increments in which the rotations exceed 10–15 degrees, or the strains exceed a few percent over important parts of the mesh (these limitations are associated with our Newton approach to solving the equations). Thus, any method that is stable for this size of increment should be satisfactory. However, I would be surprised to see conditionally stable algorithms that can handle such increments: typical yield strains in metals are about 10^{-3} , so that the increment size is about ten times the size of the yield surface in strain space, and I assume that the stability limit of a conditionally stable operator would not be larger than the yield strain. Because we have found the full Newton method to be effective, we want an algorithm that is sufficiently simple so that the Jacobian matrix, $(\partial \mathbf{T} / \partial \mathbf{E})_{t+\Delta t}$, where \mathbf{T} is the stress, \mathbf{E} is the strain, and $t + \Delta t$ is the time at the end of the increment (where we write the equilibrium equations in an implicit code), can be obtained exactly for common plasticity models. It is desirable that this matrix have an additional property: several plasticity models of practical importance are derived from a locally smooth potential which is also the yield surface, so that the rate equations of the model give a symmetric form $\partial \mathbf{T} / \partial \mathbf{E}$. We would like the integration operator to preserve this symmetry—this is of practical significance with respect to computational cost. The operator we use in Abaqus does not do so, except in a restricted class of plasticity models (which does not include some of those that are available the code and which have a symmetric rate form).

We first integrate the kinematic aspect of the model by using the algorithm of Hughes and Winget (1980) to define an increment of rotation, which we apply to all vector and tensor valued functions at the material point; we also follow the Hughes-Winget suggestion and define the strain increment by the central difference approximation. Various authors have commented on the inadequacies of this class of method in the context of theories that involve tensor valued hardening variables, such as classical kinematic hardening theory. I do not see much utility in simple kinematic hardening with respect to finite strain applications, and the approach does not suffer the same obvious deficiency when it is used with

the isotropic yield models that we generally use for metals and soils at finite strains. The algorithm leads to a symmetric initial stress matrix, which is tedious to derive.

We are left with the integration of the change of state associated with deformation. The following discussion considers only isothermal, rate independent behavior with isotropic hardening. In Abaqus we use the same approach for rate dependent models (except where we integrate explicitly, as mentioned in Section 2.2 above), for non-isothermal cases (including fully coupled temperature-displacement calculations) and for more complicated hardening models, such as kinematic hardening.

The isotropic hardening plasticity models in Abaqus all have the following structure. We assume a strain rate decomposition,

$$d\mathbf{E} = d\mathbf{E}^{el} + d\mathbf{E}^{pl}$$

where $d\mathbf{E}$ is a differential change in total mechanical strain, $d\mathbf{E}^{el}$ is a differential change in the elastic strain, and $d\mathbf{E}^{pl}$ is a differential change in the inelastic (plastic) strain. We have introduced the Hughes-Winget approach to account, in an approximate way, for the rigid-body rotation of the material during the increment, and to define a finite increment of total strain during the increment. This allows us to write this decomposition in integrated form as

$$\mathbf{E} = \mathbf{E}^{el} + \mathbf{E}^{pl} \quad (1)$$

where \mathbf{E} , \mathbf{E}^{el} and \mathbf{E}^{pl} are defined as summations of the corresponding rotated values at the start of each increment and the incremental values associated with that increment. During the constitutive calculations \mathbf{E} is known, except in the case of plane stress.

We assume an elastic strain energy potential so that the stress, \mathbf{T} , is defined as

$$\mathbf{T} = \frac{\partial W}{\partial \mathbf{E}^{el}} \quad (2)$$

where $W = W(\mathbf{E}^{el}, \theta)$ is the strain energy potential, and θ is the temperature.

Some of the plasticity models that we use in Abaqus assume linear elasticity, while others (soils models) use a nonlinear elasticity in which the logarithm of the pressure stress is proportional to the volumetric strain. None of these elasticity models has internal constraints such as incompressibility, so that (2) defines the stress completely. It can be differentiated to give

$$\partial \mathbf{T} = \left[\frac{\partial^2 W}{\partial \mathbf{E}^{el} \partial \mathbf{E}^{el}} \right] : \partial \mathbf{E}^{el}$$

which we write as

$$\partial \mathbf{T} = \mathbf{D}^{el} : \partial \mathbf{E}^{el} \quad (3)$$

A simple isotropic hardening model has the yield function

$$f = 0 \quad (4)$$

where $f = f(\mathbf{T}, \theta, H^\alpha)$, with H^α being a set of hardening parameters; f is defined such that, whenever $f < 0$, the response is purely elastic. The models we use in Abaqus all have a smooth yield function, so that $\partial f / \partial \mathbf{T}$, $\partial^2 f / \partial \mathbf{T} \partial \mathbf{T}$, and $\partial f / \partial H^\alpha$ are well defined everywhere on f .

The flow rule is written

$$d\mathbf{E}^{pl} = d\lambda \frac{\partial g}{\partial \mathbf{T}} \quad (5)$$

where $d\lambda$ is a positive scalar and $g(\mathbf{T}, \theta, H^\alpha)$ is the flow potential. Since the material models being discussed are rate independent, $d\lambda$ is determined only by the kinematic solution at the material point being considered.

The hardening parameters evolve with plastic strain:

$$dH^\alpha = h^\alpha(d\mathbf{E}^{pl}, \mathbf{T}, \theta, H^\beta)$$

where h^α defines the hardening: h^α must be homogeneous of degree one in $d\mathbf{E}^{pl}$ for the model to be rate independent. Therefore

$$dH^\alpha = d\lambda h^\alpha\left(\frac{\partial g}{\partial \mathbf{T}}, \mathbf{T}, \theta, H^\beta\right) \quad (6)$$

The plasticity model is now defined, except for choosing particular forms for the elastic strain energy potential, W , the yield function, f , the flow potential, g , and the hardening rules, h^α .

The only rate equations in the formal definition of the material model are the evolutionary rule for the hardening and the flow rule. The simplest operator which may at least fulfill the requirement of unconditional stability mentioned at the beginning of this section is the backward Euler method, which can be introduced into (5) to give

$$\Delta \mathbf{E}^{pl} = \Delta \lambda \frac{\partial g}{\partial \mathbf{T}} \quad (7)$$

while (6) is written

$$\Delta H^\alpha = \Delta \lambda h^\alpha\left(\frac{\partial g}{\partial \mathbf{T}}, \mathbf{T}, \theta, H^\beta\right) \quad (8)$$

In these equations and in all of the following, all quantities are evaluated at the end of the time increment.

Remark: Ortiz and Popov (1984) propose the midpoint rule as a more accurate operator, based on an error analysis with small strain increments. Their analysis appears to assume that the material is yielding during the entire increment. This is often not the case—in most calculations there are always material points where yielding begins part way through the increment. At such points the mid-point rule requires solution for the initial intersection with the yield surface during the increment, and application of the rule only within the

yielding part of the increment. This creates some formidable algebra, especially if one wishes to derive the Jacobian. Our analysis of this problem leads us to the conclusion that, even for an associated flow material, the Jacobian will, in this case, not be symmetric, and that its non-symmetric part may not be very small. We choose the backward Euler method partly because it does not introduce this difficulty, and so retains sufficient tractability that we can complete the algebra rather easily. It is interesting to note that Ortiz and Popov confirm the conclusion of Schreyer et al. (1979): that, for strain increments that are not small compared to the size of the yield surface in strain space, the backward Euler method exhibits the highest accuracy of any of the simple methods that they tested.

From a computational viewpoint the problem is now algebraic: we must solve the nonlinear equations (1), (2), (4), (7) and (8), and thus define the state of the material at the end of the increment. The "material stiffness matrix",

$$[\mathbf{D}] = \left[\frac{\partial \mathbf{T}}{\partial \mathbf{E}} \right]$$

must also be defined for the overall Newton scheme that we use for the equilibrium equations. Simple manipulations lead to the definition

$$[\mathbf{D}] = [[\mathbf{I}] + \Delta\lambda[\mathbf{H}] : [\mathbf{L}]]^{-1}[\mathbf{H}]$$

where $[\mathbf{I}]$ is the fourth order identity tensor, and

$$[\mathbf{H}] = [\mathbf{D}]^{el} : [\mathbf{Z}]$$

$$[\mathbf{Z}] = [\mathbf{I}] - \frac{1}{d} \hat{\mathbf{N}} \mathbf{M} : [\mathbf{D}]^{el}$$

$$\hat{\mathbf{N}} = \mathbf{N} + \Delta\lambda \frac{\partial \mathbf{N}}{\partial H^\alpha} c_{\alpha\beta} h^\beta$$

$$\mathbf{N} = \frac{\partial g}{\partial \mathbf{T}}$$

$$c_{\alpha\beta} = \left(\delta_{\alpha\beta} - \Delta\lambda \left(\frac{\partial h^\alpha}{\mathbf{N}} : \frac{\partial \mathbf{N}}{\partial H^\beta} + \frac{\partial h^\alpha}{\partial H^\beta} \right) \right)^{-1}$$

$$d = \mathbf{M} : [\mathbf{D}]^{el} : \hat{\mathbf{N}} - \frac{\partial f}{\partial H^\alpha} c_{\alpha\beta} h^\beta$$

$$\mathbf{M} = \frac{\partial f}{\partial \mathbf{T}} + \Delta\lambda \frac{\partial f}{\partial H^\alpha} c_{\alpha\beta} \left(\frac{\partial h^\beta}{\partial \mathbf{N}} : \left[\frac{\partial \mathbf{N}}{\partial \mathbf{T}} \right] + \frac{\partial h^\beta}{\partial \mathbf{T}} \right)$$

and

$$[\mathbf{L}] = \frac{\partial \mathbf{N}}{\partial \mathbf{T}} + \Delta\lambda \left(\left[\frac{\partial \mathbf{N}}{\partial H^\alpha} c_{\alpha\beta} \frac{\partial h^\beta}{\partial \mathbf{N}} \right] : \left[\frac{\partial \mathbf{N}}{\partial \mathbf{T}} \right] + \frac{\partial \mathbf{N}}{\partial H^\alpha} \frac{\partial h^\alpha}{\partial \mathbf{T}} \right)$$

For a non-associated flow material, $\partial f / \partial \mathbf{T}$ and \mathbf{N} are not related, so that $[\mathbf{D}]$ cannot be symmetric. For an associated flow material, $\partial f / \partial \mathbf{T}$ and \mathbf{N} (and hence $\hat{\mathbf{N}}$) will be co-linear. Therefore $[\mathbf{D}]$ will be symmetric if

$$\frac{\partial h^\alpha}{\partial \mathbf{N}} : \left[\frac{\partial \mathbf{N}}{\partial \mathbf{T}} \right] = 0 \quad \text{and} \quad \frac{\partial h^\alpha}{\partial \mathbf{T}} = 0$$

which is the case, for example, for the simple Prandtl-Reuss model that is commonly used for metals, but is not true for the modified Cam-clay model that is sometimes used for clays. However, in associated flow cases when it is not true, the non-symmetric contributions to $[\mathbf{D}]$ appear only in the terms multiplied by $(\Delta\lambda)^2$, so that they are of the order of (the plastic strain increment)² compared to unity. This suggests that, for practical purposes, the lack of symmetry should not degrade the convergence of the Newton iterations for equilibrium if we approximate $[\mathbf{D}]$ with its symmetric part, and our experience to date confirms this—at least, the performance of the algorithm with the symmetric approximation has been satisfactory, by our standards.

There remains the problem of solving the algebraic equations for the state at the end of the increment. In the general case, this is not a simple matter: whatever method is used should not be expensive on the computer, but it must work for the tightly curved yield functions that appear in some models of practical interest, even when the strain increment is many times the typical size of the yield surface in strain space. The problem is made more awkward in cases such as Cam-clay, where exponential terms appear in the elasticity and in the hardening. Our approach has been as follows. A Newton scheme should work, provided we choose suitable variables as a basis, and provided we start with a good guess. The plastic strain increments, $\Delta \mathbf{E}^{pl}$, should be a suitable set of variables. Then Newton's method for (1), (2), (4), (7) and (8) is the linear equations

$$[[\mathbf{I}] + \Delta\lambda[\mathbf{Z}] : [\mathbf{L}] : [\mathbf{D}]^{el}] : \mathbf{C}^{pl} = [\mathbf{Z}] : (\Delta\lambda \mathbf{N} - \Delta \mathbf{E}^{pl}) + \hat{\mathbf{N}} \frac{1}{d} f$$

which are solved for \mathbf{C}^{pl} , the improvement to the solution for $\Delta \mathbf{E}^{pl}$:

$$\Delta \mathbf{E}^{pl} = \Delta \mathbf{E}^{pl} + \mathbf{C}^{pl}$$

The elastic strain is then obtained from (1), the stress from (2). $\Delta\lambda$ from the projection of (7) onto \mathbf{N} :

$$\Delta\lambda = \frac{\mathbf{N} : \Delta \mathbf{E}^{pl}}{\mathbf{N} : \mathbf{N}}$$

and ΔH^α from (8).

This loop is repeated until the components of $\Delta \mathbf{E}^{pl}$ converge. A tight convergence criterion is necessary, so that the solution is accurately defined: this is essential for the overall Newton scheme for the equilibrium equations to converge quadratically.

The Newton method requires solution of the linear system at each iteration. Although the system of equations is not large (at most six), these computations are done at each integration point and at each iteration of the overall equilibrium solution, so that it is desirable to solve the problem more efficiently than by direct application of the Newton method. In addition, when the strain increment is large, a reasonable starting guess is necessary for the method to converge. For these reasons we have been using a projection of the problem onto a smaller number of variables to start the solution. If this subspace is chosen appropriately, we should be able to develop a useful estimate of the solution at low cost. The general idea is as follows.

Let $\tilde{\mathbf{K}}^\alpha$, $\alpha = 1, \dots, n$, be a set of tensors that are orthonormal:

$$\tilde{\mathbf{K}}^\alpha : \tilde{\mathbf{K}}^\beta = \delta^{\alpha\beta}$$

and choose n to be less than the number of stress components in the actual problem.

The $\tilde{\mathbf{K}}^\alpha$ are chosen for a particular guess and are fixed during the solution for that guess.

Assume that the plastic strain increment is

$$\Delta \mathbf{E}^{pl} = \Delta e_\alpha^{pl} \tilde{\mathbf{K}}^\alpha \quad (9)$$

The elasticity, (2), and the strain rate decomposition, (1), then define the stress, which we require to satisfy yield, (4). The integrated flow equation, (7), is imposed in the sub-space:

$$\Delta \mathbf{E}^{pl} = \Delta \tilde{\lambda} \tilde{\mathbf{N}}$$

where

$$\tilde{\mathbf{N}} = \tilde{\mathbf{K}}^\alpha \tilde{\mathbf{K}}^\alpha : \mathbf{N} \quad (10)$$

Since the $\tilde{\mathbf{K}}^\alpha$ are orthonormal, (9) and (10) then define

$$\Delta e_\alpha^{pl} = \Delta \tilde{\lambda} N^\alpha \quad (11)$$

where

$$N^\alpha = \tilde{\mathbf{K}}^\alpha : \mathbf{N}$$

The Newton scheme described above for the full stress space projects directly:

$$[\delta^{\alpha\beta} + \Delta \tilde{\lambda} Z^{\alpha\gamma} (\hat{\mathbf{K}}^\gamma : [\mathbf{L}] : [\mathbf{D}]^{el} : \tilde{\mathbf{K}}^\beta)] c^\beta = Z^{\alpha\beta} (\Delta \tilde{\lambda} N^\beta - \Delta e_\beta^{pl}) + \hat{N}^\alpha \frac{f}{\tilde{d}}$$

where

$$Z^{\alpha\beta} = \delta^{\alpha\beta} - \frac{1}{d} \tilde{\mathbf{K}}^\alpha : \hat{\mathbf{N}} \mathbf{M} : [\mathbf{D}]^{el} : \tilde{\mathbf{K}}^\beta$$

$$\hat{N}^\alpha = \tilde{\mathbf{K}}^\alpha : \hat{\mathbf{N}}$$

and

$$\tilde{d} = \mathbf{M} : [\mathbf{D}]^{el} : \tilde{\mathbf{K}}^\alpha \tilde{\mathbf{K}}^\alpha : \hat{\mathbf{N}} - \frac{\partial f}{\partial H_\alpha} c_{\alpha\beta} h^\beta$$

These linear equations provide the c^α , the improvements to the Δe_α^{pl} :

$$\Delta e_\alpha^{pl} = \Delta e_\alpha^{pl} + c^\alpha$$

At each iteration the solution is updated as described for the full stress-space solution, except that $\Delta \tilde{\lambda}$ is calculated by projecting (11) onto N^α :

$$\Delta \tilde{\lambda} = \frac{N^\alpha \Delta e_\alpha^{pl}}{N^\beta N^\beta}$$

The utility of this technique depends on the choice of the $\tilde{\mathbf{K}}^\alpha$, and on n . Clearly it is unlikely that it would be worthwhile to use $n > 2$. Most of the plasticity models in Abaqus are isotropic, in the sense that the yield function, f , and the flow potential, g , are defined in terms of the stress invariants. An obvious choice for the $\tilde{\mathbf{K}}^\alpha$ in these cases is $n = 2$, with

$$\tilde{\mathbf{K}}^1 = \frac{1}{\sqrt{3}} \mathbf{I}$$

(here \mathbf{I} is the identity matrix),

and

$$\tilde{\mathbf{K}}^2 = \sqrt{\frac{3}{2}} \frac{\mathbf{S}^0}{q^0}$$

where \mathbf{S}^0 is the deviatoric part of \mathbf{T}^0 , the stress that would occur at the end of the increment if there were no plasticity occurring in the increment, and

$$q^0 = \sqrt{\frac{3}{2} \mathbf{S}^0 : \mathbf{S}^0}$$

If the only stress dependencies in f and g are the effective pressure stress,

$$p = -\frac{1}{3} \mathbf{I} : \mathbf{T}$$

and the deviatoric stress magnitude, q ; and the elasticity is isotropic, (and the hardening is isotropic, as has been assumed here), the subspace solution is the exact answer to the

problem, except for the plane stress case. For some simple yield surfaces and hardening definitions the subspace problem can be solved in closed form, without the need for iteration. For the simplest case of a Mises material, where q is the only stress term in f and g , the one-dimensional sub-space defined by $\tilde{\mathbf{K}}^2$ above provides the exact solution. For perfect plasticity the method is then precisely Wilkins' "radial return" algorithm.

Our experience with more general yield functions and flow potentials, where the third stress invariant is also used, is that the two-dimensional sub-space provides a satisfactory guess, from which the solution can be completed usually with no more than two iterations of the full stress-space problem.

6. Closure

The paper has presented a brief review of our experience in the recent past in the general area of nonlinear structural analysis of metal shells. It is not difficult to identify the most severe limitations in the methods we have used so far, to write down a "wish list" of areas where we would like to be using better methods than those we have used to date. The list is as follows.

- We do not like the rapid deterioration in the accuracy of the isoparametric shell elements that we currently use when they are not rectangles and the shell is thin.
- Reduced integration elements are attractive because they minimize constitutive calculations, which are often a significant part of the computational cost when the material model is not simple. Numerical artifices, such as hourglass control, are not objectionable, when they are effective and are well understood. Our own understanding so far is lacking.
- The characteristic hinging of thin metal shells under compressive loads raises difficulties and opportunities. The difficulties are associated with concerns about capturing this behavior with smooth discretizations. The opportunities are indicated by the remarkable success of the rigid-plastic models of Wierzbicki. Perhaps it is true that, in practical cases, load bearing members are too thick for this to be important, although movies of front-end collision experiments on cars shown very pronounced hinging.
- A straightforward finite membrane strain shell formulation would have wide application in several problem areas that we often encounter. The axi-symmetric formulation we have been using is based substantially on the assumptions introduced by Rodal in his thesis, although we have taken a rather different approach in detail. We do not have a good appreciation of the limitations of the formulation.
- We expect to continue to work with implicit methods in many problem areas, including a large part of the analysis work associated with structural design. If this is a correct assessment, it would be highly desirable to have equation solution methods that are more effective and less difficult to use than Newton's method. This is, from our point of view, one of the most severe limitations that we face in practical applications.
- So much shell behavior involves branching on the solution path that it would be very satisfying to have methods that can handle this unassisted.

- Finally, we have not mentioned rezoning, or the treatment of localization, but these are becoming important issues in practical cases.

7. References

T. Belytschko and C.-S. Tsay, "A Stabilization Procedure for the Quadrilateral Plate Element with One-Point Quadrature", *Int. J. Num. Meth. Eng.*, Vol. 19, pp. 405–419, 1983.

H.M. Hilber, T.J.R. Hughes and R.L. Taylor, "Collocation, Dissipation and Overshoot for Time Integration Schemes in Structural Dynamics", *Earthquake Eng. and Struc. Dyn.*, Vol. 16, pp. 99–117, 1978.

T. J. R. Hughes and J. Winget, "Finite Rotation Effects in Numerical Integration of Rate Constitutive Equations Arising in Large Deformation Analysis", *Int. J. Num. Meth. Eng.*, Vol. 15, pp. 1862–1867, 1980.

B. M. Irons, "The Semi-Loof Shell Elements", Chapter 11 of *Finite Elements for Thin Shells & Curved Members*, edited by D. G. Ashwell and R. H. Gallagher, John Wiley & Sons, London, 1976.

B. M. Irons and S. Ahmad, *Techniques of Finite Elements*, Ellis Horwood Ltd., Chichester, England, 1980.

B. M. Irons and M. Loikkanen, "An Engineers' Defense of the Patch Test", *Int. J. Num. Meth. Eng.*, Vol. 19, pp. 1391–1401, 1983.

M. Ortiz and E. P. Popov, "Accuracy and Stability of Integration Algorithms for Elastoplastic Constitutive Relations", Brown University, Division of Engineering Report, 1984.

K. C. Park and G. M. Stanley, "A Curved C^0 Shell Element Based On Assumed Natural-Coordinate Strains", Applied Mechanics Lab., Lockheed Palo Alto Research Lab., Palo Alto, Cal., 1984.

E. Riks, "An Incremental Approach to the Solution of Snapping and Buckling Problems", *Int. J. Solids and Struc.*, Vol. 15, pp. 329–551, 1979.

J. J. A. Rodal and E. A. Witmer, "Finite-Strain Large-Deflection Elastic-Viscoplastic Finite-Element Transient Response Analysis of Structures", contractor report 159874, NASA Lewis, 1979.

J. C. Simo and M. Ortiz, "A Unified Approach to Finite Deformation Elastoplastic Analysis Based on the Use of Hyperelastic Constitutive Equations", Brown University Division of Engineering Report, 1984.

G. Strang, and G. J. Fix, *An Analysis of the Finite Element Method*, Prentice-Hall, New Jersey, 1973.

H. L. Schreyer, R. F. Kulak and J. M. Kramer, "Accurate Numerical Solutions for Elastic-Plastic Models", *Trans. ASME, J. Pressure Vessel Technology*, Vol. 101, 1979.

P. S. Symonds, "Survey of Methods of Analysis for Plastic Deformation of Structures under Dynamic Loading", Brown University Division of Engineering Report. 1967.

COMPUTERIZED STRUCTURAL MECHANICS FOR 1990'S: ADVANCED AIRCRAFT NEEDS

A. V. Viswanathan and B. F. Backman
Boeing Military Airplane Company
Seattle, Washington

Abstract

This paper describes the needs for computerized structural mechanics (CSM) as seen from the standpoint of the aircraft industry. It projects these needs into the 1990's with special focus on the new advanced materials. It identifies the major areas:

- preliminary design/analysis
- research
- detail design/analysis

and elaborates on the role of local/global analyses in these different areas.

The lessons learned in the past are used as a basis for the design of a CSM framework that could modify and consolidate existing technology and include future developments in a rational and useful way.

A philosophy is stated, and a set of analyses needs driven by the emerging advanced composites is enumerated. The roles of NASA, the universities, and the industry are identified.

Finally, a set of rational research targets is recommended based on both the new types of computers and the increased complexity the "industry" faces.

Computerized structural mechanics should be more than new methods in structural mechanics and numerical analyses. It should be a set of engineering applications software products that combines innovations in structural mechanics, numerical analysis, data processing, search and display features, and recent hardware advances and is organized in a framework that directly supports the design process.

There are two aspects to the development of engineering applications software: (1) innovations and (2) improvements in the productivity of engineers.

While this paper concentrates on the first aspect, it is still important to consider the productivity aspect in the development of new methods because the software ultimately will be part of the set of tools used in the design of aircraft structures.

Structural Analysis Needs for the Aircraft Industry

- **New capabilities**
 - **Concepts**
 - **Materials**
 - **Details**
- **Improved productivity**
 - **Engineering workstations**
 - **Tutorial software**
 - **CAD/CAM interfaces**

New non-ductile materials have entered the scene and will be part of the picture for a long time to come. These materials require more detailed analyses not only because of micro-considerations and interlaminar effects but also because of their unforgiving nature which entails a significant participation of "secondary" effects in the failure modes. The complex definition of these materials and their responses required to identify behavior has resulted in a large increase in the information volumes for processing of a typical aircraft problem in the structures field.

At the same time, we have seen tightening up of requirements not only in terms of adverse environment but also in terms of improved quality. In both cases this tightening results in a need for more sophisticated analyses.

The general areas of computing technology and information science have seen dramatic changes in both hardware and software. There have been hardware changes that could be parlayed into: (1) optimization of structures with practical constraints, (2) nonlinear analyses at reasonable cost, and (3) micro-analyses at acceptable storage requirements. Developments have occurred in the field of data processing that make data base management a natural cornerstone in the future of CSM. Finally, there have been developments in the fields of graphics and CAD/CAM which would make the establishment of standardized user interfaces extraordinarily useful both from the standpoint of technology and management.

Motivation for Increased Development

- **Advanced composites**
 - **Higher complexity**
 - **More unforgiving**
 - **Larger information volumes**
- **More stringent requirements**
 - **Lighter weights**
 - **Higher temperatures**
 - **Longer lives**
- **Improved computing technology**
 - **Numerical methods**
 - **Data processing/search and display**
 - **CAD/CAM**

The only result of significance in the aircraft industry is to produce a new "better" aircraft. Any other result is intermediate and will only be acceptable if it contributes to this improvement. The success of CSM is therefore contingent on three abilities:

- To promote and support new technology in the fields of structures, numerical methods, and data processing
- To include and improve methods in the preliminary design and analysis of new aircrafts (the need to develop and refine conceptual methods for the selection and comparisons of sophisticated candidates with no experience basis should be a strong driver in goal setting)
- To introduce new methods and integrate these methods in a framework that can produce the visibility, data reduction, and sophistications that are necessary to support a production effort in a new era

What Constitutes Success?

➡ Better aerospace vehicles

➡ Therefore "CSM" must

- Support technology development
- Provide NEW preliminary design and analysis methods
- Automate detail design and analysis methods

The successful development of new methods in computerized structural mechanics requires an understanding of the psychology of the situation.

1. An efficient method that supports the needs of the industry must be established.
2. A practical input and output language must be used.
3. A set of useful user interfaces must be available.
4. An appropriate amount of visibility and data reduction features should be provided.
5. A realistic marketing effort should be launched.
6. A developer and user interaction is required.

Yardstick for Success for CSM

**Computerized methods development
will not be successful unless
the new “product” is totally acceptable
to the users**

The CSM framework should be designed and implemented with common design questions in mind. Of these, the question of material choice and mix is very complex. An optimum distribution of materials in the primary structure of an airplane is dependent on the critical failure modes. Is it strength (in tension), stability, fatigue, damage tolerance, or stiffness requirements that size a local detail? Again, it can be seen how a global/local/global cycle must be used to come to grips with this design problem. All the design drivers are closely related, and the question of environment involves the determination of temperature, moisture, presence of chemical, corrosion, and risks of FOD*. These effects need to be assessed at least on a parallel track. The performance requirements, such as speed, load factors, roll rates, and landing speeds, all are very basic and need to be addressed again in a local/global fashion. Finally, design criteria are "soft" early in the design process and evolve in a cyclic manner as the local/global analyses of the structure mature.

What are the Structural Design Drivers?

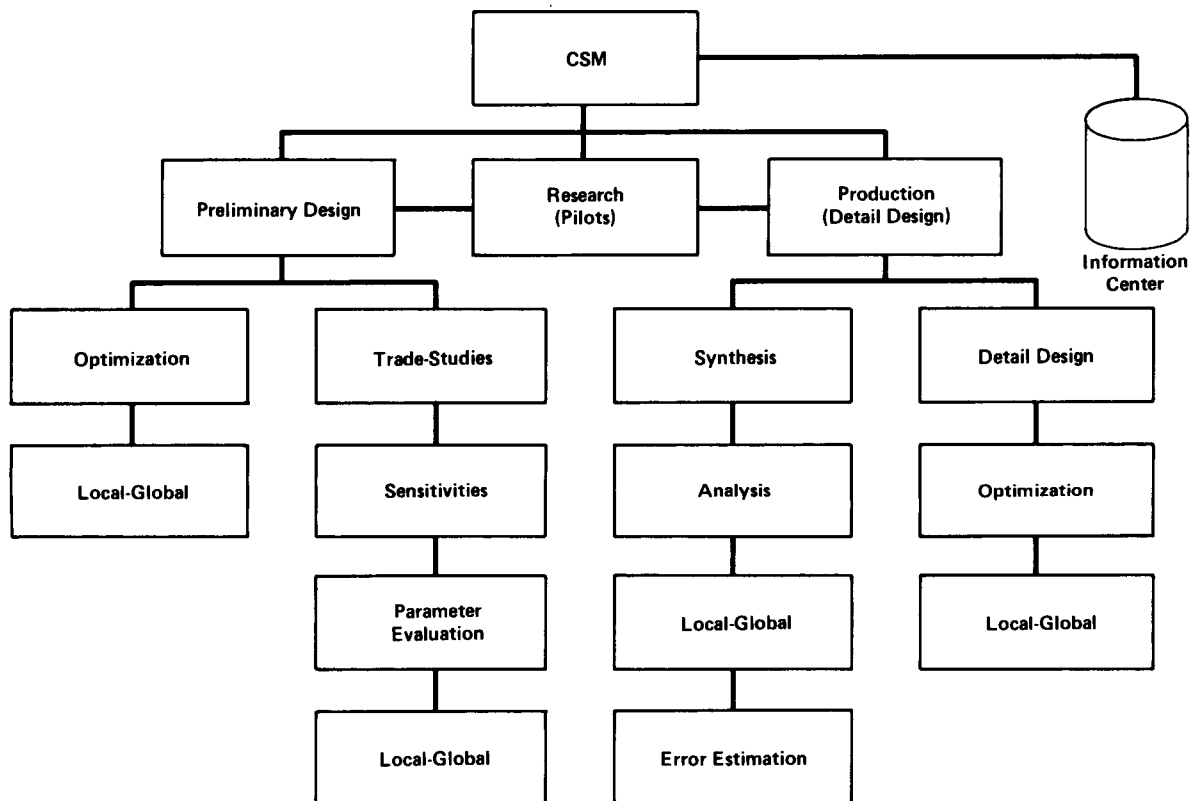
- **Material choice and mix**
 - **Strength**
 - **Stiffness**
 - **Damage tolerance**
 - **Fatigue performance**
- **Environment**
- **Performance requirements**
- **Design criteria**

*foreign object damage

There are three fields of structures that have to be included in order to give attention to all the aspects of engineering design in the aircraft industry. The first of these fields is the research activity that leads to a new technology and a better understanding of the ingredients in the design process. The second field is the preliminary design and analysis that leads to the selection of the appropriate family of vehicles or products. Both syntheses and analyses on one side and parameters and variables on the other are involved in this design and analysis.

The third field is the detail design and analysis, of which perhaps the local/global concept is more applicable than anywhere else. Here the objective is to learn as much as possible about a specific design candidate. The demands are especially stringent on CSM to include features that facilitate data reduction and display as well as sophisticated local analyses methods.

CSM, What Should be Included?



The overwhelming requirement in the preliminary design and analysis phase deals with synthesis methods that discriminate between parameters and variables in the design process and makes it possible to study alternative formulations.

Preliminary Design and Analysis

- **Access to expert system data base (trade study results)**
- **Parametric results from local-global analyses as baseline for optimization**
- **Determination of sensitivities (data base)**
- **Algorithms for parameter determinations and variable selections**
- **Strategy algorithms, history function**

The role of research in the CSM arena is central and its primary purpose is to promote better understanding of the materials and structures technologies required to produce better vehicles. To that end, it is essential to have significant development in (1) structural and continuum mechanics, (2) numerical methods, and (3) failure prediction and test evaluation. This must be done in a framework that allows for empiricism that is sensitive to user needs, aims at synthesis, and produces pilot capabilities with well-defined interfaces.

Research

- **Phenomena evaluation for better understanding**
- **Special purpose detail analyses**
- **New numerical methods**
- **“Pilot” development**
- **Test data evaluation methods**
- **Empirical corrections methods**

The detail design and analysis fields have always been characterized by large volumes of data. This is becoming more and more the case as the new advanced composite materials are introduced into the production environment. (Just consider the simple problem of calculating margins of safety.) This together with more complex requirements and the less forgiving nature of the materials has resulted in order-of-magnitude increases in data volumes in addition to a myriad of local analyses methods demands.

Production (Detail Design and Analysis)

- **Postprocessing, search and display (graphics)**
- **Strength checking (software for detail analyses)**
- **Management visibility and access**
- **Data base access (parametric representation...allowables)**
- **Automated resize (optimization), baseline updates**
- **Strategy for local-global analyses**
- **Error estimation**

The CSM framework has to have an information center that supports the local/global analyses and makes it possible to combine all aspects of theory, empiricism, test data, and criteria.

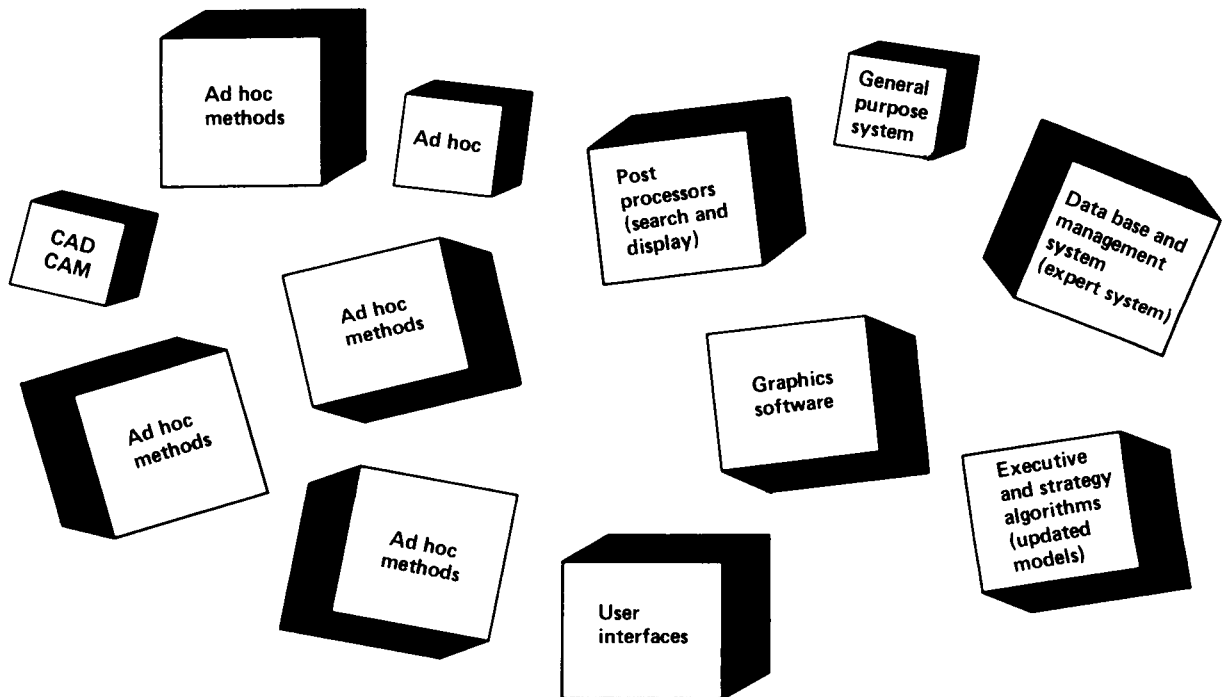
This data base should be such that it supports the preliminary design, detail design, and research.

Information Center (Data Base)

- **Self educating expert system (better modeling, improved parameter evaluation, trade projections)**
- **Test data evaluation and empirical correction**
- **Sensitivities**
- **History writing**
- **Search and display**

The building blocks for CSM are naturally many and of very different nature, but the central core in a local/global system will have to be a set of special purpose methods that in a very flexible way can be included, updated, and replaced.

What are the Building Blocks?



From the standpoint of the aircraft industry, the last 20 years have seen an increased reliance on highly specialized experts to perform special purpose (local) analyses. This is an undesirable development, and the CSM framework should provide the requirements necessary to curtail and reverse this software engineering trend. At the same time a number of general systems have emerged, reached maturity, expanded to a high level of maintenance budget needs, and then stagnated. Finally, the framework must provide local/global communications and user interfaces.

Need from Users

- **Present situation**
 - **Increasing dependence on highly specialized experts**
 - **General purpose systems in use require more user driven long term plans**
 - **As the present trend shows high maintenance burden and early stagnation**
 - **Proper framework for global to multi-local communications is missing**

The strategy for the long-term CSM development could include: (1) identification of existing software that can be modified for a new environment, (2) a plan that is based on an evaluation of industry needs and experience with new operating systems, compilers, hardware, and methods, (3) the new hardware potential that must be part of the picture when targeting special fields; optimization is still a cost/hardware constrained activity, and (4) emphasis that should be based on the priorities that come out of industry needs. NASA has a key role to play in pulling all these together.

Strategy

- **Plan that modifies and uses existing software where feasible**
- **Plans driven by what the industry presently knows**
- **Plan that recognizes new hardware potentials**
- **Plan that encourages more “applied research” into fields driven by industry needs**

The requirements mentioned previously translate into a set of organizational issues that are essential to CSM development follow-through. First, the size of the software demand is such that a national commitment is necessary if the technology benefits are to be realized. Second, a number of general-purpose scientific systems and an assortment of special-purpose software have demonstrated the need for a framework designed to draw upon the benefits of both. Finally, user acceptance will depend on good visibility and easy access to solutions and results.

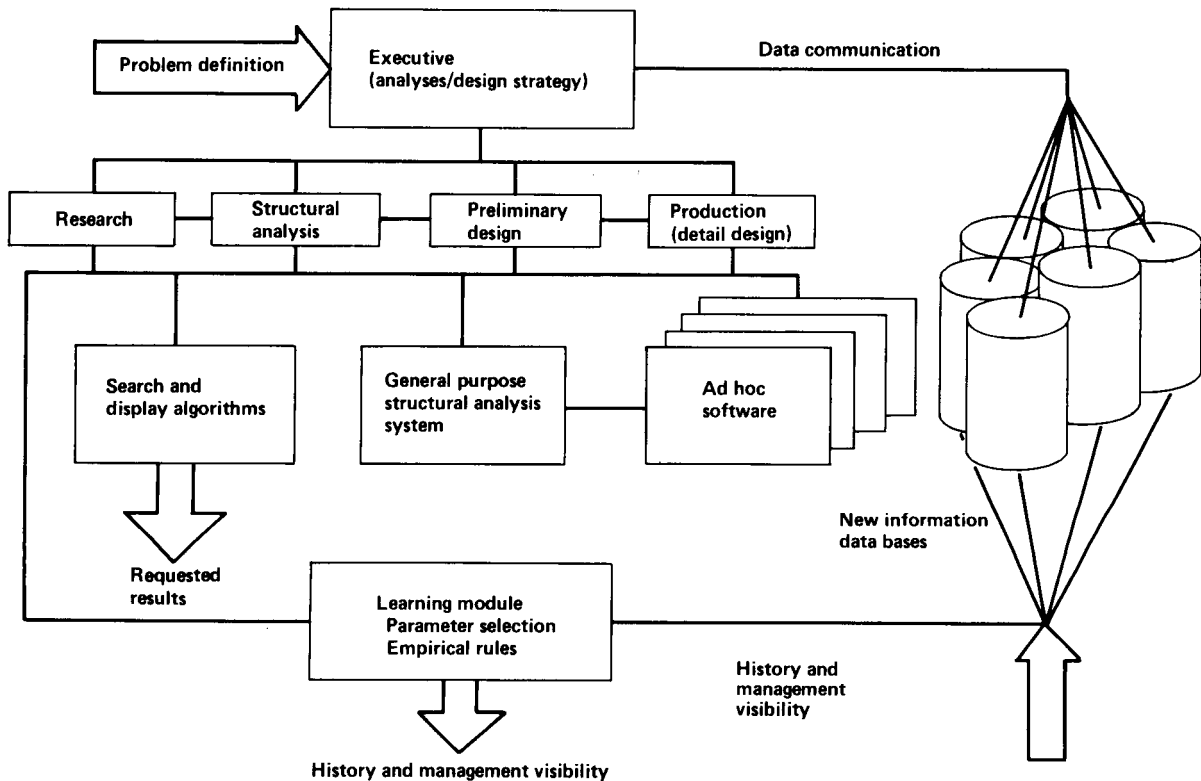
Fundamental Organizational Issues

- **The successful development of CSM requires central leadership**
- **CSM should provide the framework for unification of general/global and special/local methods**
- **Search and display algorithms and standardized user interfaces are as important as solutions**

The backbone of the CSM design approach is the data management, but the foundation for success can be found in the quality of the special-purpose methods and the associated software. Experience has shown that the success and acceptance of these methods are very dependent on availability of required data, ease of interpretation of results, and visibility of the steps leading to the final solution.

The approach should also deal with the present trends in structures, in which on one hand we are moving into fields with very limited experience bases and on the other hand we produce innovation through point design testing. We can address both situations in a framework that not only accepts new methods with ease but also uses these methods for experience development and empirical evolution.

Could This be a “Better” Approach?



The typical situation in the structures field involves a number of analyses that are directed at different levels of resolution. These levels will have to be revisited a number of times during the design process. The evolution of the design, therefore, obviously requires a number of global-to-local-to-global transitions involving huge data volumes. The efficiency of these transitions requires as much attention as the methods development.

Local-Global Analyses

- **Internal loads/global, overall FE analyses, stability, aeroelastic effects, flutter, ...**
- **Detail stress analysis (linear/nonlinear), “local” buckling, allowables processing, postbuckling analyses, interlaminar analyses, residual strength, damage tolerance analyses, fatigue analysis**
- **Local optimization, multilevel optimization, automated remodeling, ...**
- **Error estimation**
- **Micro-detail-macro strategy (transitions)**

An understanding of the design drivers leads to the identification of a number of fields in which work should be done in order to efficiently produce preliminary and final designs. Here the preliminary and final design processes are different in focus, but similar principles could serve both enterprises and emphasis should be given to synthesis-like features.

Development Support

- **Determination of material development targets**
- **Selection of “best” structural materials (based on most important design drivers, and determine failure progression)**
- **Establishing environmental effects**
- **Determining an optimum set of performance requirements (temperature/speed, weight, load factor,...)**
- **Exploration of sensitivities to variation in design criteria**
- **Multilevel strength, stress/stability analyses**
- **Multilevel optimization**
- **Pre- and postprocessing with search and display features**

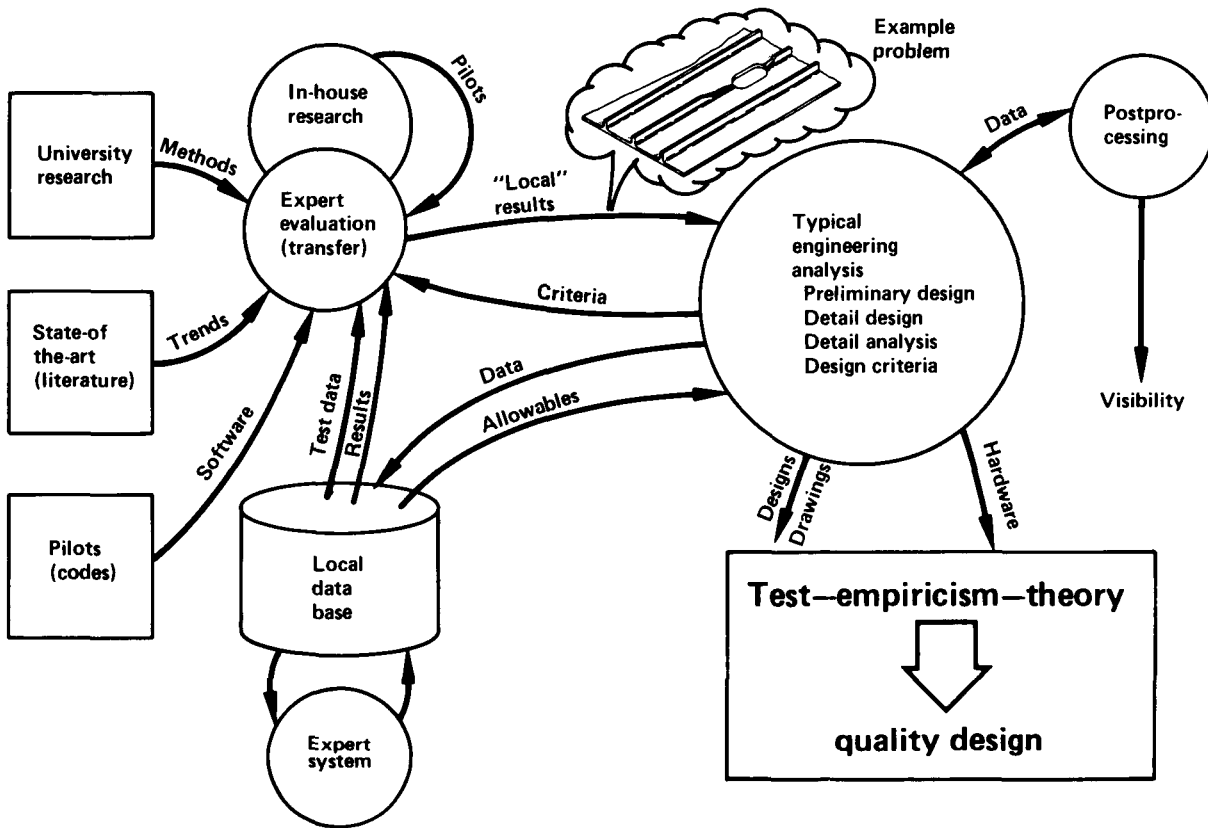
The framework for CSM must be such that both categories of users/developers (typical engineers and experts) can be allowed their proper influence in a manner that promotes natural roles and allows for organized communications of both needs and results.

What are We Aiming For?

- **A system for technology development and improved scientific understanding for experts**
- **A system for preliminary design, detail design and analyses for typical engineers in production environments**

The local/global analysis development is seen as a three-pronged effort that includes: (1) advanced methods, (2) typical engineering analyses, and (3) data base and associated methods for experience development. The advanced methods would primarily be intended for expert evaluations, but could, if properly packaged, be included among the typical engineering analyses. The expert evaluations would produce direct input to the overall design effort, but would also feed the data base and indirectly support both parametric evaluations and preliminary design approaches. The example problem represents a number of analyses that belong on the local level and should be considered from the design standpoint.

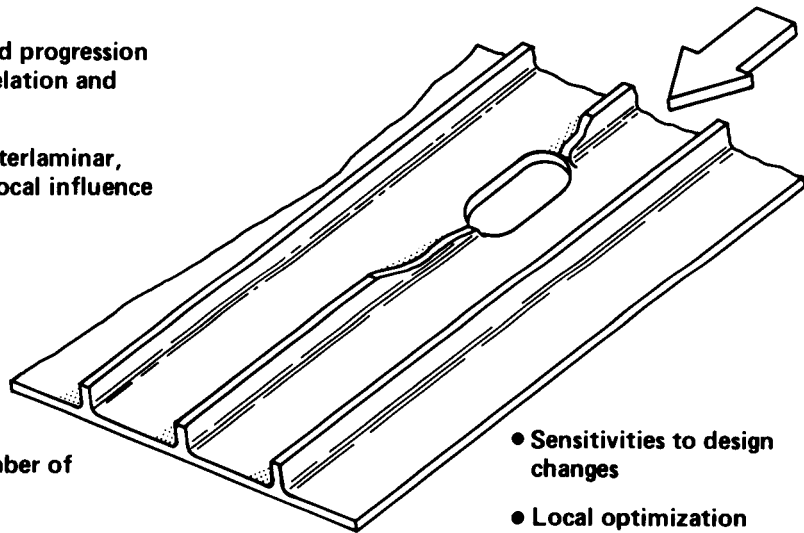
Local-Global Analysis



The example problem in the context shown in the previous figure represents a number of analyses and syntheses. All of these belong to the local or micro level when seen from the standpoint of the design process. All of these, and a number of others, are natural development (research) targets that after implementation should either directly feed results into the "typical" engineering activity or produce results to the local data base for future use or parametric access. All these developments should result in pilot capabilities or products that in an ad hoc fashion would satisfy one of the above two requirements.

What are the Analyses for the Example Problem?

- Local failure modes and progression based on test, for correlation and empirical inputs
- Multilevel analyses: interlaminar, local damage growth, local influence of postbuckling
- Stability analyses
- Postbuckling analyses
- Nonlinear material response,...
- Flaw-growth for a number of damages
- Allowables determination
- Parametric representation and empirical input to data base



- Sensitivities to design changes
- Local optimization
- Automated "remodeling" and transition regions
- Screening for criticalities, critical requirements
- "Gross" FE properties
- Residual stresses

The example problem is one of many analogous local problems that must be solved as part of the design process. Each one of these problems is solved a number of times during the evolution of the design. We are therefore forced to minimize the number of micro evaluations and expert involvement in order to produce an efficient design process. This can be done by including the results on a communal experience basis in a way that supports the practical engineering tasks. Many of the local analysis packages must be designed for minimum involvement by experts. In many types of problems, it will be necessary to identify important parameters and produce solutions in advance in a format that can be accessed in a data base management environment. Very similar considerations apply to the allowables question, whether it is processing of material structural allowables. Finally, in the failure prediction development, one can see elements of the other types and here one can hardly expect practical support of the design process without a two-level software development that ultimately depends on empirical modifications.

The Example Problem

- **Local with “micro” requirements**
- **Local analyses with minimum expert requirements**
- **Basis for parameter selection**
- **Panel for allowables generation**
- **Candidate for failure prediction development**

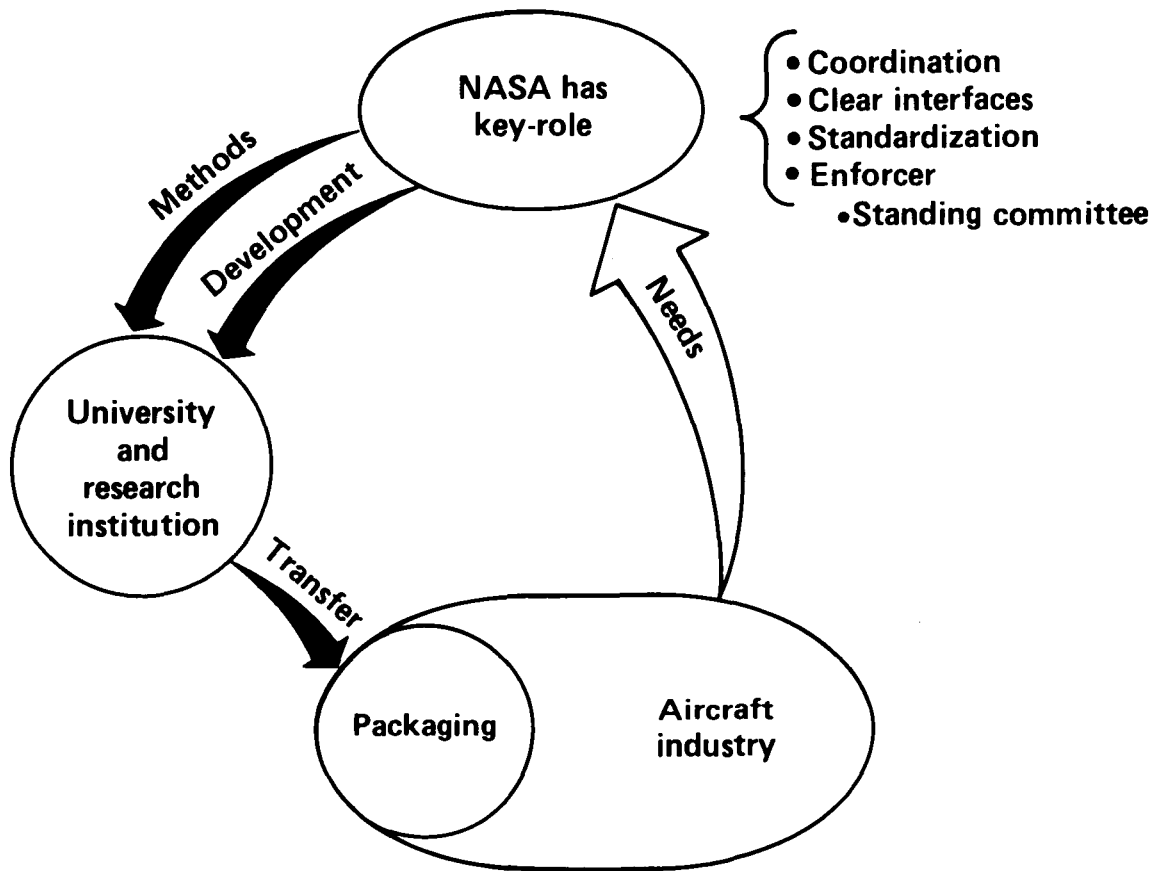
In conclusion, we find that there are a number of research targets waiting for initiative from the engineering community. It has become clear, however, that the majority of these targets will be missed if not approached from the standpoint of their overall role in the design process. It is also felt that the objectives will not be met without a proper technology transfer to the users. This naturally involves both the software packaging and the promotional activity necessary for dissemination.

Conclusions

- **Research should include**
 - **Analysis methods (nonlinear, advanced materials,... structures)**
 - **Numerical methods**
 - **Artificial intelligence**
 - **Search and display/data processing**
- **Emphasis on technology transfer to industry**

NASA has a key role in the development of CSM for the 1990's and beyond. A national effort is required if new materials (composites), new computers, new methods, and new requirements are to be addressed in a manner that establishes economic advances and preserves the superior safety record established by the aircraft industry. NASA has a role not only in leadership but also as communicator assuring technology transfer and promoting user acceptance.

Recommendations



NONLINEAR RESPONSE OF A BLADE-STIFFENED GRAPHITE-EPOXY PANEL
WITH A DISCONTINUOUS STIFFENER: WORK IN PROGRESS

Norman F. Knight, Jr., William H. Greene,
and W. Jefferson Stroud
NASA Langley Research Center
Hampton, Virginia

INTRODUCTION

Discontinuities and eccentricities are usually present in practical structures. In addition, potential damage of otherwise perfect structures is often an important design consideration. Predicting the structural response in the presence of discontinuities, eccentricities, and damage is particularly difficult when the component is built from graphite-epoxy materials or is loaded into the nonlinear range. Recent interest in applying graphite-epoxy materials to aircraft primary structures has led to several studies of postbuckling behavior and failure characteristics of graphite-epoxy structural components (e.g., refs. 1-3). However, these studies concentrated on two topics: prediction of the overall response of composite structural components in the postbuckling range or failure mechanisms and analytical failure prediction techniques for fibrous composite materials. The problem of calculating detailed stress distributions around discontinuities in buckled, composite structural components for use with the various analytical failure prediction techniques has not been thoroughly explored.

The purpose of this paper is the application of computational methods to the detailed stress analysis problem which is the focus of this session of the workshop. One approach to uncovering the difficulties of this type of analysis and to providing specific directions for future research in this area is a direct attack on the problem using currently available analysis tools. A candidate problem has been selected and the remainder of the paper describes experiences from calculating its structural response.

PRECEDING PAGE BLANK NOT FILMED

BLADE-STIFFENED GRAPHITE-EPOXY PANEL WITH A DISCONTINUOUS STIFFENER:
FOCUS PROBLEM

The focus problem for the local/global stress analysis session of this workshop is to determine the nonlinear response of a flat blade-stiffened graphite-epoxy panel with a discontinuous stiffener. The material system for the panel is T300/5208 graphite-epoxy with a nominal ply thickness of 0.0055 in. Typical lamina properties for this graphite-epoxy system are 19,000 ksi for the longitudinal Young's modulus, 1,890 ksi for the transverse Young's modulus, 930 ksi for the shear modulus, and 0.38 for the major Poisson's ratio. The panel skin has 25 plies $([+45/0_2/\bar{+45}/0_3/\pm 45/0_3/\bar{+45}/0_3/\pm 45/0_2/\bar{+45}])$ and the blade stiffeners have 24 plies $([+45/0_2/\bar{+45}])$. The overall length of the panel is 30 in., the overall width is 11.5 in., stiffener spacing is 4.5 in., stiffener height is 1.4 in., and the hole diameter is 2 in. The loading is uniform axial compression. The loaded ends of the panel are clamped and the sides are free.

This problem was selected as the focus problem because experimental results are available and because it has characteristics which often require a local/global analysis. These characteristics include a discontinuity, eccentric loading, large displacements, large stress gradients, high inplane loading, and a brittle material system. This problem represents a generic class of laminated composite structures with discontinuities in which the interlaminar stress state becomes important.

- Graphite-epoxy (T300/5208)
- Flat panel with three blade stiffeners
- 30 in. long
- 11.5 in. wide
- Stiffener spacing of 4.5 in.
- Stiffener height of 1.4 in.
- 2.0-in.-diameter hole
- 25-ply panel skin
- 24-ply blade stiffeners
- Axially loaded with loaded ends clamped and sides free



ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

LOCAL/GLOBAL TERMINOLOGY

The definition of a local/global structural analysis is not precise. For example, several levels of detail are considered in the analysis of an aircraft structure, and the concepts of local and global can change with every change in analysis level. To some, the entire aircraft is the global model, and a fuselage section is the local model. To others, lamination theory represents the global model, and micromechanics is used for the local model. For this workshop, a local/global analysis is a local, detailed stress analysis within a larger, less-refined global analysis model. The overall response of the panel is the global problem; the response near the hole is the local problem.

Current areas of research associated with local/global methodologies are described in the literature (e.g., refs. 4 and 5). One research area (discretization procedures) includes finite-element methods, finite-difference methods, and boundary element or boundary integral methods. Adaptive mesh refinement, h- and p-convergence, and error analysis are current research topics in this area which address discretization effects in the presence of a large stress gradient. A second area (refined theories) includes research in transverse shear formulations and three-dimensional elasticity solutions. These topics focus on the mathematical representation of the mechanics of the problem. A third area includes classical and closed-form solutions which are often restricted to simple geometries, specific boundary conditions and material systems, and often to a linear response prediction. A fourth area is hybrid techniques in which two or more methods are used simultaneously but in different domains of the structure. All four of these areas of research are addressed in the local/global session of this workshop.

- **Concept of local/global changes with analysis level**

- **Definitions**

- Global means overall panel response
- Local means response near the hole

- **Local/global methodologies**

- Discretization procedures
- Refined theories
- Classical and closed-form solutions
- Hybrid techniques

APPLICATION OF THE FINITE-ELEMENT METHOD
TO LOCAL/GLOBAL STRESS ANALYSIS

The local/global methodology adopted for this paper is the finite-element method because of its generality. The first step in applying the method was to develop and verify a finite-element model of the focus problem. Model verification involved solving simpler example cases and comparing the results with other analytical results. This model verification process was aided by the development of a flexible mesh generation capability which allowed various finite-element discretizations to be evaluated rapidly and systematically. The mesh generation capability also provided an easy way to construct and study several idealized example cases.

Once an adequate finite-element model for the global response was verified, the nonlinear structural response was calculated. To identify the local modeling detail required to predict accurately the stress distribution near the hole, linear stress analyses were performed on a rectangular plate with a circular hole using several refined 2-D models near the hole.

- Finite-element model development

- Finite-element model verification

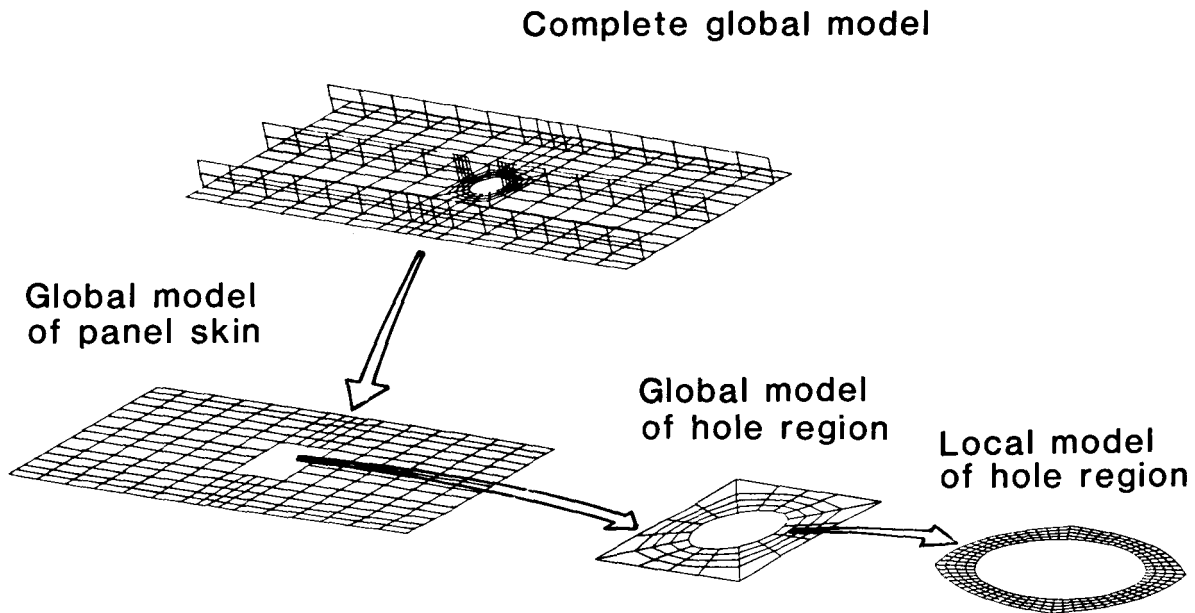
- Global nonlinear response prediction

- Local linear stress analysis

FINITE-ELEMENT MODEL DEVELOPMENT

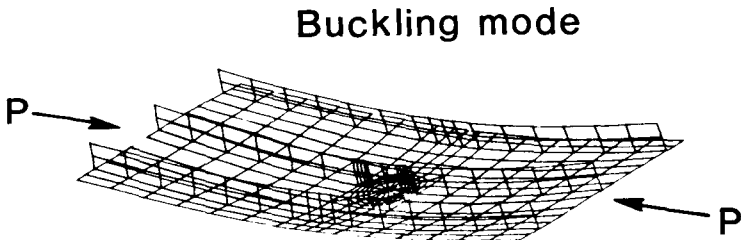
The model development strategy is to predict the global nonlinear response using the complete model and then to construct a refined, local 2-D model for a small distance away from the hole to predict accurately the large stress gradient. Displacements and rotations from the global nonlinear solution obtained using the complete model will be applied to the refined model and the state of stress will be determined. This strategy will be referred to as a multi-level or "zoom-in" approach.

The automated mesh generation capability allowed versatile modeling of the complete problem as well as local regions near the hole. The analyst could specify the number of elements across the stiffener depth and down the length of the panel, the number of rings of elements around the hole, and the number of elements around the hole, and could control the element spacing in the vicinity of the hole. Models could also be generated with the hole and discontinuous stiffener filled-in or with no stiffeners.



FINITE-ELEMENT MODEL VERIFICATION:
BLADE STIFFENED PANEL

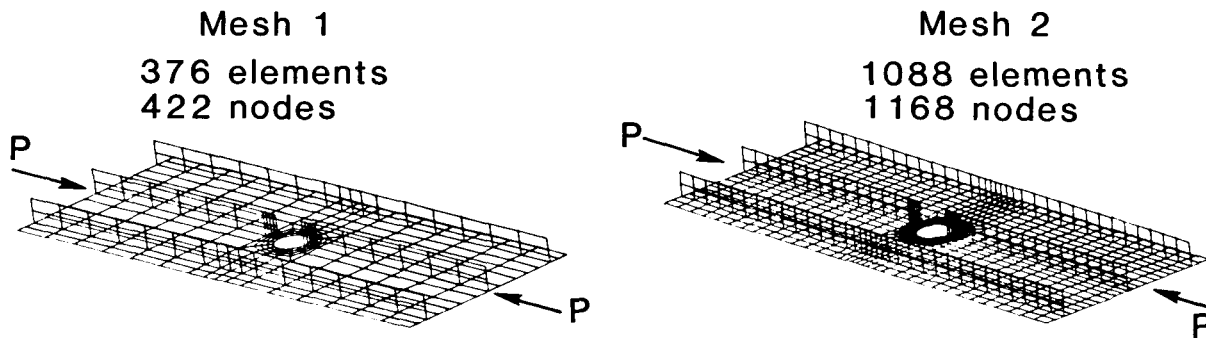
In the first step of the verification process, a simplified version of the focus problem was studied. This simpler problem was identical to the focus problem except that the hole and discontinuous stiffener are filled-in and the end boundary conditions are now simple support conditions. For this prismatic panel, an exact solution was obtained using the PASCO computer code (Panel Analysis and Sizing COde, ref. 6). The finite-element analysis system EAL (Engineering Analysis Language, ref. 7) was used for the finite-element analysis. The finite-element model used in the verification was developed from that of the focus problem to determine if any problems related to element distortion or aspect ratio were present. The prebuckling boundary conditions and end loading are such that a uniform stress state is present in the skin and the blade stiffeners. The three lowest buckling eigenvalues obtained using EAL are very close to the PASCO solutions.



	Eigenvalues		
	1	2	3
PASCO	44536	51063	61601
EAL	44652	51182	60975
Difference	+0.26%	+0.23%	-1.02%

FINITE-ELEMENT MODEL VERIFICATION:
FOCUS PROBLEM

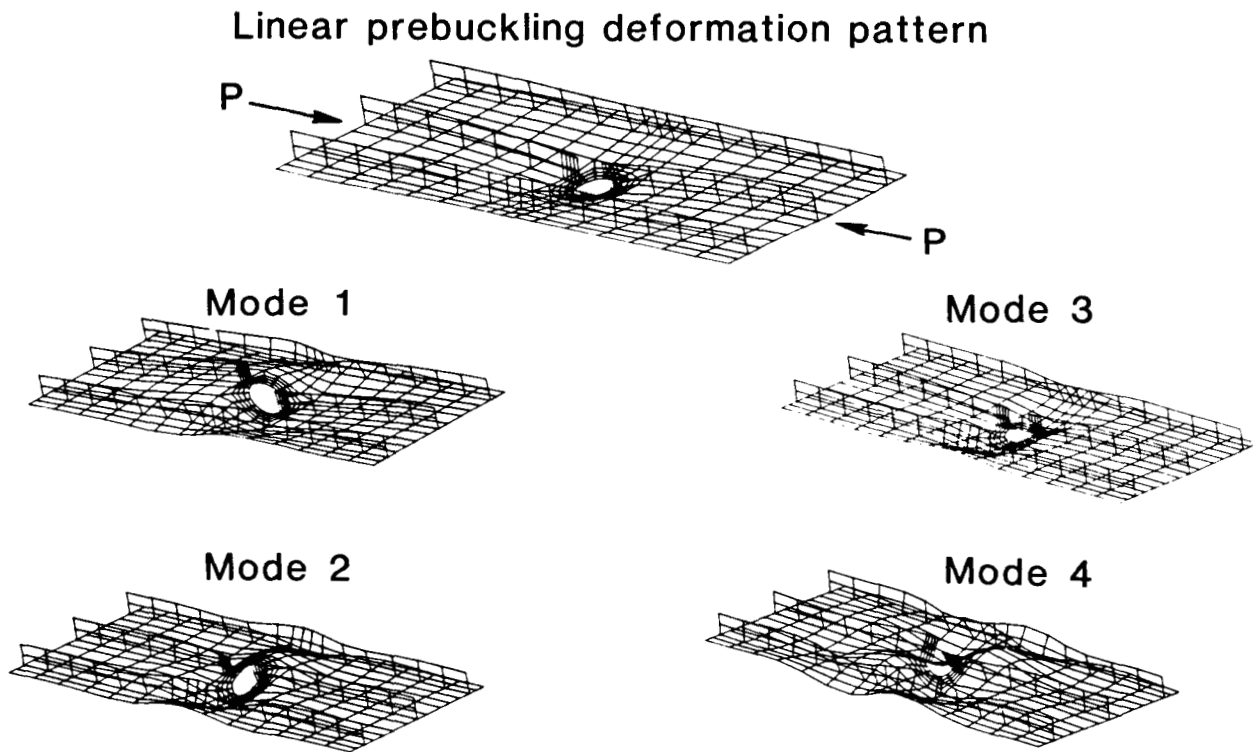
The next step of the verification process was to define an adequate finite-element model for the global response of the focus problem. Finite-element model verification for the focus problem started with a "reasonable grid" of 376 4-node assumed-stress quadrilateral elements and 422 nodes. This discretization is referred to as Mesh 1. A second, refined grid of 1088 4-node quadrilateral elements and 1168 nodes (Mesh 2) was generated for model verification. Linear bifurcation buckling solutions for Mesh 1 and Mesh 2 were compared to establish the adequacy of the models. The three lowest eigenvalues from both discretizations agree within approximately one percent.



	Eigenvalues			
	1	2	3	4
Mesh 1	41378	52754	54288	55344
Mesh 2	41829	52533	54259	56895
Change	-1.08%	+0.42%	+0.05%	-2.73%

LINEAR BIFURCATION BUCKLING SOLUTION

Oblique views of the prebuckling deformation pattern and the eigenmodes corresponding to the four lowest eigenvalues are shown in this figure. The discontinuous stiffener leads to an eccentric loading condition which causes large out-of-plane displacements to develop near the hole from the onset of loading. Because of this coupling between inplane and out-of-plane displacements, no linear equilibrium path exists and the linear bifurcation buckling results do not have the traditional meaning. The linear buckling solutions may be used as a guide in selecting the initial load for the nonlinear analysis and in choosing a load step size. However, their main use is in studying the effects of spatial discretization.

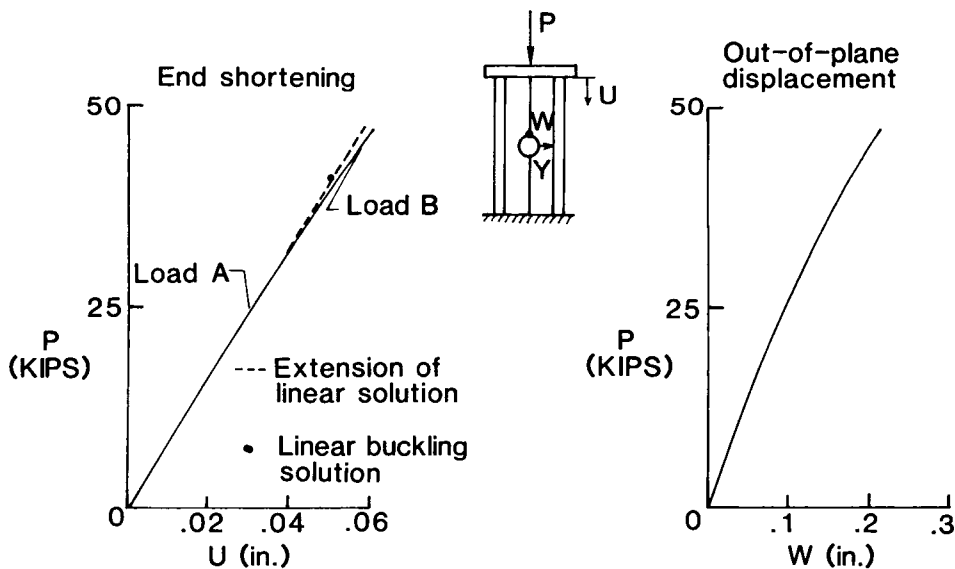


GLOBAL NONLINEAR RESPONSE PREDICTION

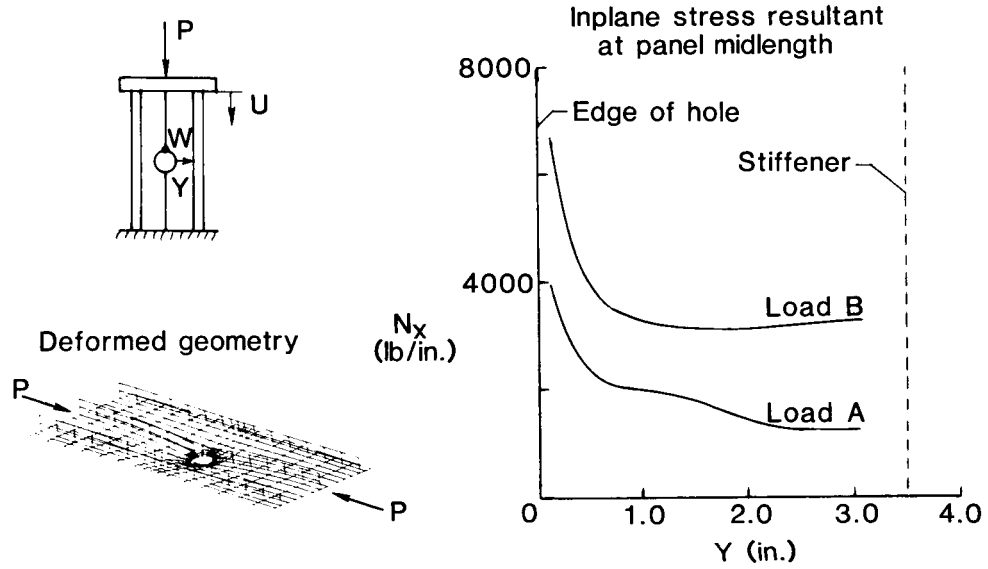
The global nonlinear response predicted for the focus problem was obtained using a new release of EAL. This new release has a nonlinear analysis capability using a corotational formulation with linear strain-displacement relations within the elements. For this problem, the loading was applied in increments with a full Newton-Raphson algorithm. Convergence was based on the maximum error in the residual force vector.

An oblique view of the deformed geometry for the last calculated solution is similar to the linear solution shown previously, indicating that the primary equilibrium path is being followed. A global response quantity, end shortening, is nearly a linear function of the applied load. A local response quantity, out-of-plane displacement at the edge of the hole and blade stiffener, indicates large displacements from the onset of loading. Longitudinal inplane stress-resultant distributions for two values of the applied load, as a function of distance from the hole, indicate high inplane stresses and a high stress gradient near the hole.

These high inplane stresses and stress gradients coupled with the large out-of-plane displacements and the free edge of the hole may cause material nonlinearities, local failures, and/or delaminations to develop in order to provide local stress relief mechanisms (like plasticity in metal structures) near the hole and blade stiffener. However, an accurate prediction of the effects of these mechanisms on the global nonlinear response is beyond the current analysis capabilities. Stress fracture criteria are developed in reference 8 for an inplane loading condition in which the influence of these stress relief mechanisms can be accounted for in failure studies without knowing exactly what is happening locally near the hole. The point stress failure criterion developed in reference 8 and applied in reference 9 to a broad class of laminated composite plates with holes will be used in this study as a guide for establishing an adequate finite-element model for predicting the stress distributions near the hole.

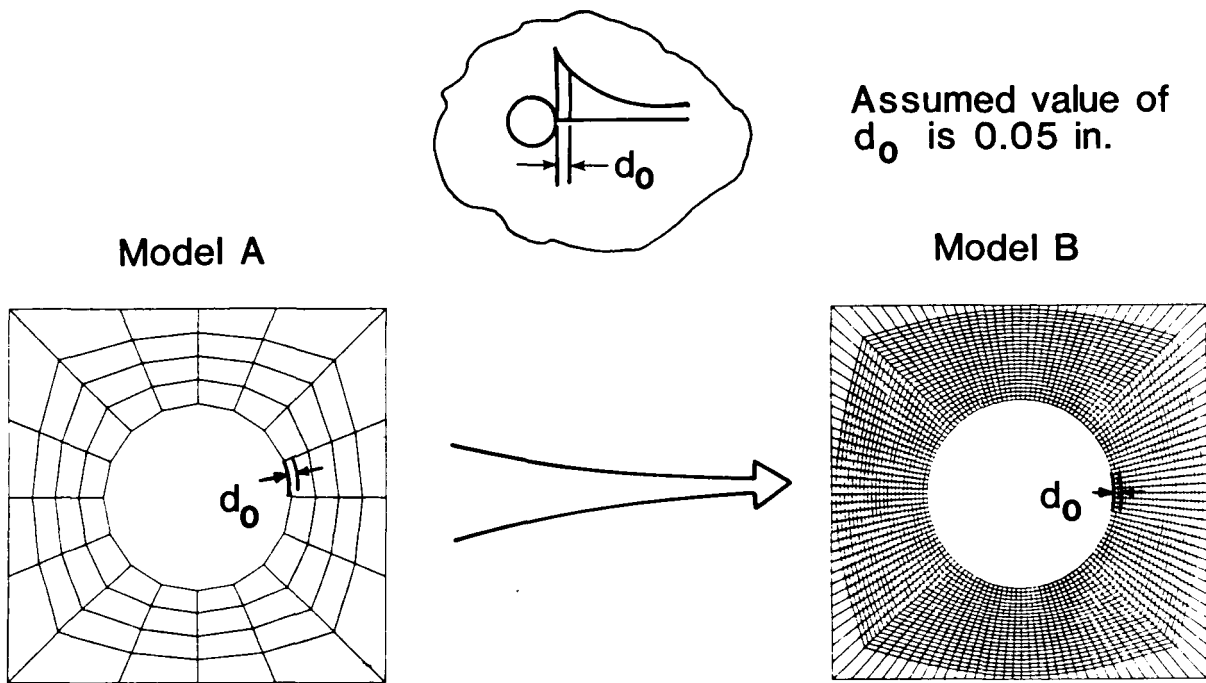


GLOBAL NONLINEAR RESPONSE PREDICTION



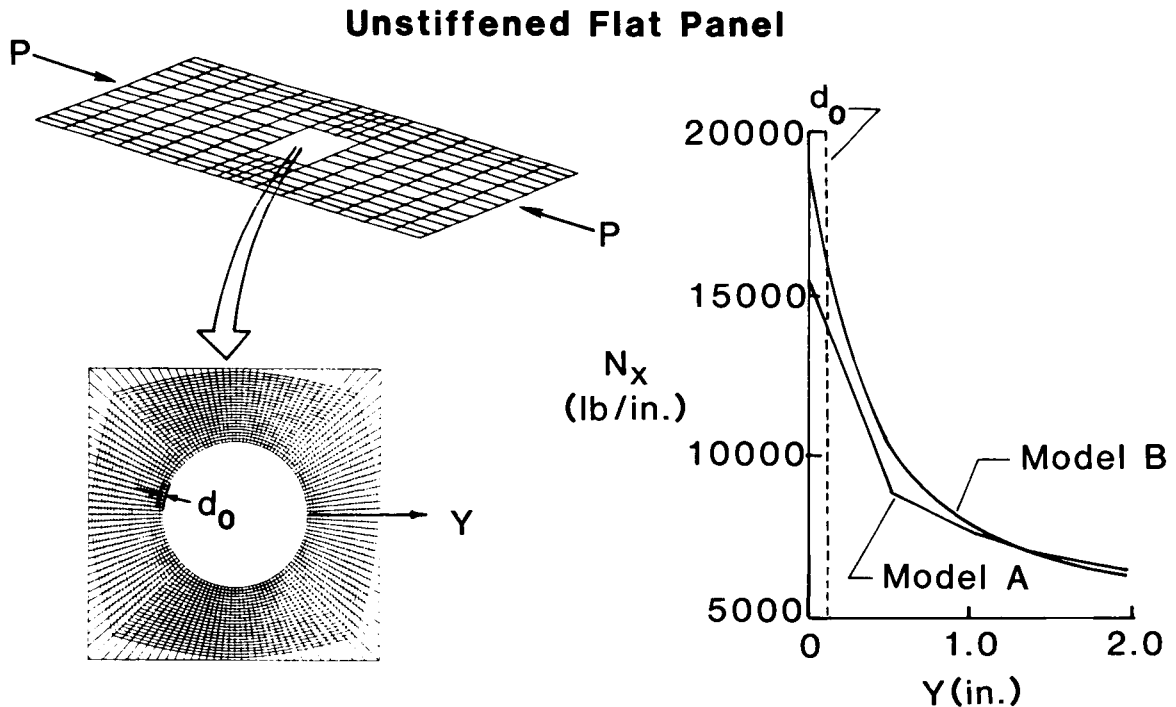
POINT STRESS FAILURE CRITERION

The point stress failure criterion assumes that failure occurs when the stress at a distance d_0 away from the edge of the hole reaches ultimate. The distance d_0 is a characteristic dimension which takes the form of a material property. A consequence of using this criterion is that an accurate prediction of the state of stress precisely at the free edge of the hole is not required. It is only necessary to have an accurate stress prediction at a distance d_0 from the edge of the hole. Based on results of reference 9 with a similar graphite-epoxy material system, a value of $d_0 = 0.05$ inches was assumed. Model A corresponds to the global model near the hole for Mesh 1. Model B is a refined model which more accurately predicts the stress gradient near the hole.



LOCAL LINEAR STRESS ANALYSIS

To understand how the spatial discretization near the hole affects the prediction of the stress at d_0 , a simpler structural configuration was considered which did not include the stiffeners. Using this planar structure, an adequate 2-D finite-element model was identified for the local stress analysis. This approach provided the necessary insight required for a multi-level model of the focus problem. An alternate approach would have been to use an adaptive mesh refinement procedure. However, no such procedure was available. The longitudinal inplane stress resultant distributions as a function of distance away from the edge of the hole are shown for two finite-element models. The results from both models approach one another away from the hole. However, at a distance d_0 from the edge of the hole, the solutions for Models A and B differ by 12.5 percent. The finite-element model in the vicinity of the hole was refined by doubling the number of elements. The inplane stress resultant at d_0 changed by only 2.2 percent between Model B and a model with half as many elements.



STATUS AND ADDITIONAL TASKS

The overall strategy for this study is to predict the global nonlinear response using the complete global model and then to construct a refined, local 2-D model for a small distance away from the hole. The global nonlinear response has been predicted for the focus problem and the local modeling detail required for an accurate local stress analysis near the hole of an unstiffened panel has been identified. The tasks that remain to be completed for the focus problem include performing the multi-level analysis and applying a failure criterion. The multi-level analysis will involve applying the displacements and rotations from the global nonlinear solution on the refined local 2-D model and determining the state of stress at d_0 . In addition, a three-dimensional model near the discontinuity will be required for an accurate determination of the through-the-thickness state of stress (i.e., normal and transverse shearing stress distributions). The use of 3-D elements within a 2-D model will also require a strategy for the transition or blending of the two models.

Status

- Finite element model developed and verified
- Global nonlinear response predicted
- Required modeling detail identified for stress gradient near the hole for an unstiffened panel

Additional tasks for focus problem

- Perform multi-level 2-D analysis (refined, local 2-D model)
- Apply point stress failure criterion
- Perform multi-level 3-D analysis (refined, local 3-D model)

SUMMARY

The local/global nonlinear stress analysis of a blade-stiffened graphite-epoxy panel with a discontinuous stiffener is indeed a computational challenge. Substantial engineering effort is required in modeling the structure, in verifying that the physics of the problem are modeled, and in interpreting the predicted nonlinear solutions. Approximately fifty percent of the analysis effort to date was devoted to model development and verification. The development of a flexible mesh generation capability was essential for model verification. Several models of similar but simpler structures were required and easily generated using the automated mesh generator.

To complete the analysis effort for the focus problem several issues need to be addressed. The transition or interface between the various levels of the multi-level model needs to be defined. An adaptive mesh refinement procedure is needed to automate the definition of the finite-element models at each stage of the multi-level approach. To obtain a detailed through-the-thickness stress distribution, a three-dimensional analysis will be required and the number of three-D elements through-the-thickness of the laminate needs to be determined. In addition, to predict the response of the structure up to overall structural failure, a progressive failure analysis capability would be required in which various failure mechanism and failure criteria are incorporated.

- Substantial engineering effort required in modeling, model verification, and response interpretation.
- Flexible mesh generation capability essential to model verification.
- Definition of transition/interface region between multi-level models required.
- Required number of 3-D elements through-the-thickness to be determined.
- Nonlinear analysis procedure with progressive failure analysis capability needed.

REFERENCES

1. Starnes, James H., Jr., Dickson, John N. and Rouse, Marshall, "Postbuckling of Graphite-Epoxy Panels," ACEE Composite Structures Technology: Review of Selected NASA Research on Composite Materials and Structures, NASA CP-2321, 1984, pp. 137-160.
2. Starnes, James H., Jr. and Williams, Jerry G., "Failure Characteristics of Graphite-Epoxy Structural Components Loaded in Compression," NASA TM-84552, September 1982. Also, in Mechanics of Composite Materials; Recent Advances, Proceedings of the IUTAM Symposium on Mechanics of Composite Materials, Z. Hashin and C. T. Herakovich (editors), Pergamon Press, Inc., 1983, pp. 283-306.
3. Noor, A. K., Shuart, M. J., Starnes, J. H., Jr., and Williams, J. G. (Compilers), Failure Analysis and Mechanisms of Failure of Fibrous Composite Structures, NASA CP-2278, 1983.
4. Zienkiewicz, O. C., "The Generalized Finite Element Method---State of the Art and Future Directions," Journal of Applied Mechanics, ASME, Vol. 50, December 1983, pp. 1210-1217.
5. Noor, A. K. and Pilkey, W. D. (editors), State-of-the-Art Surveys on Finite Element Technology, ASME, New York, 1983.
6. Anderson, Melvin S. and Stroud, W. Jefferson, "A General Panel Sizing Computer Code and Its Application to Composite Structural Panels," AIAA Journal, Vol. 17, No. 8, August 1979, pp. 892-897.
7. Whetstone, W. D., EISI-EAL Engineering Analysis Language Reference Manual - EISI-EAL System Level 2091, Engineering Information Systems, Inc., San Jose, CA, July 1983.
8. Whitney, J. M. and Nuismer, R. J., "Stress Fracture Criteria for Laminated Composites Containing Stress Concentrations," Journal of Composite Materials, Vol. 8, July 1974, pp. 253-265.
9. Mikulas, Martin M., Jr., "Failure Prediction Techniques for Compression Loaded Composite Laminates with Holes," Selected NASA Research in Composite Materials and Structures, NASA CP-2142, 1980, pp. 1-34.

COMPUTATIONAL PROCEDURES FOR POSTBUCKLING OF COMPOSITE SHELLS

G. M. Stanley and C. A. Felippa
Lockheed Palo Alto Research Laboratory
Palo Alto, California

SUMMARY

A recently developed finite-element capability for general nonlinear shell analysis, featuring the use of three-dimensional constitutive equations within an efficient resultant-oriented framework, is employed to simulate the postbuckling response of an axially compressed composite cylindrical panel with a circular cutout. The problem is a generic example of modern composite aircraft components for which postbuckling strength (i.e., fail-safety) is desired in the presence of local discontinuities such as holes and cracked stiffeners. While the computational software does a reasonable job of predicting both the buckling load and the qualitative aspects of postbuckling (compared both with experiment and another code) there are some discrepancies due to (1) uncertainties in the nominal layer material properties, (2) structural sensitivity to initial imperfections, and (3) the neglect of dynamic and local material delamination effects in the numerical model. Corresponding refinements are suggested for the realistic continuation of this type of analysis.

§1. INTRODUCTION

Advanced composite materials, due to their superior strength-to-weight ratios and stiffness tailorability, have become key ingredients in the design of modern aerospace vehicles. However, the complex structural response associated with such materials coupled with the intricacy of their fabrication creates harsh requirements for numerical simulation.

Specifically, a problem that is of current interest to NASA/LaRC is the determination of the postbuckling strength of thin laminated composite shells comprising the "skin" of stiffened air-transport fuselages [1]. These shells are required to maintain safe load-carrying capability substantially beyond the point at which skin buckling (i.e., *wrinkling*) occurs. To complicate matters, aircraft fuselages typically feature local discontinuities, such as fasteners, stiffeners and *cutouts*, which can induce high local stress gradients that tend to *delaminate* the composite material. In the presence of buckling, these local delaminations can propagate throughout — and hence fail — a composite structure.

It is NASA's ultimate goal to be able to predict such phenomena analytically. To this end, they have asked us to match some experimental data obtained for a

representative test specimen [1].* We have completed the first phase of the *global* analysis, which is described in the following sections. It employs a recently developed finite-element shell analysis capability, featuring the use of three-dimensional constitutive equations within an efficient resultant-oriented framework, to determine the buckling load and explore the postbuckling regime of an elastically intact composite model. The more ambitious *global/local* analysis, which involves the prediction of local material delamination and its interaction with global structural response to determine *postbuckling strength*, is still in the planning phase.

§2. PROBLEM DESCRIPTION

§2.1 Setup

The focal problem mentioned above is depicted in Figure 1. Shown is a moderately deep (55.6 deg) cylindrical panel with a circular cutout. The panel has a 14 in. square planform, a 15 in. radius of curvature and a radius-to-thickness ratio of 150. The hole is centrally located and measures 2 in. in diameter. The composite shell wall consists of 16 layers of unidirectional graphite fibers in epoxy-resin. Each layer is .0056 in. thick (for a total of .086 in.) and the layers are arranged in the symmetric, *quasi-isotropic*** stacking sequence: $\{\pm 45/90/0/0/90/\mp 45\}$ degrees — repeated twice. The orthotropic-elastic material properties for each layer are listed in the figure. Note that these properties represent *nominal* values and will require some adjustment in the sequel (see §5).

Surrounded by a metallic test frame, the appropriate boundary conditions for the cylindrical panel are (i) fully clamped on the bottom edge, (ii) clamped except for axial motion on the top edge and (iii) simply supported on the vertical edges.

§2.2 Experimental Results

The test conducted by NASA consisted of statically imposing a uniform end-shortening, δ , to determine the load-carrying capability of the panel beyond the initial buckling load. Experimental results are shown in Figure 2.

Figure 2a represents a normalized “load versus end-shortening” curve. Note that buckling occurs abruptly, followed by a rapid drop in the axial load. Somewhere

* This problem was first suggested by Dr. Norman F. Knight, Jr. at NASA Langley Research Center. We amusingly refer to it as “Knight’s problem” both as an acknowledgment to the originator and as a reminder of the many pitfalls obstructing its numerical solution.

** The term *quasi-isotropic* refers to the fact that the resultant constitutive matrix is essentially isotropic, due to a balanced sequence of fiber angles through the thickness. However, in contrast to truly isotropic materials, there is some additional coupling between bending and twisting deformations.

between the top and bottom of this vertical branch, which spans a period of *milliseconds* in the experiment, delamination occurs near the hole (Fig. 2b). Although the delamination gets progressively worse and eventually distorts the hole, there is a secondary (postbuckling) stiffening branch in the load/displacement curve. The test was stopped at the point on this secondary branch labeled “collapse”, at which point extensive delamination is evident (see [1] for details).

§3. COMPUTATIONAL APPROACH

§3.1 Formulation

The formulation of the governing equations and associated solution algorithms used to analyze the above problem is outlined in Figure 3. A detailed description of this approach may be found in [2]. Briefly, it employs *continuum-based* (CB) shell elements, similar to the Ahmad element [3], but extended to the nonlinear regime and reduced to an economical *resultant* form in which element stiffness and force operators are *pre-integrated* through the thickness.

Shell Equations

Thickness pre-integration of the CB shell equations is achieved by augmenting the standard, Mindlin-type [4] hypotheses of *straight normals* and *zero normal stress* with two additional hypotheses, namely: *small transverse-shear strains* and *mild taper*. As argued in [2], these additional hypotheses do not significantly alter the range of applicability of the original formulation. The resulting theory, which is expressed in terms of stress-resultants rather than pointwise (continuum) stresses, is referred to as a *continuum-based resultant* (CBR) shell formulation.

Note that unlike earlier efforts to pre-integrate the CB shell equations, which typically assume a constant through-thickness variation of the surface metric (*e.g.*, [5]), the present CBR formulation bypasses this assumption and hence is not restricted to very thin shells.

Shell Elements

To spatially discretize the CBR shell equations, a variety of shell finite elements have been implemented within the above framework. However, on the basis of the numerical evaluation conducted in [2], only the following two shell elements were considered for the present analysis: (i) the nine-node Heterosis (HET) element [6], and (ii) a new nine-node assumed natural strain (ANS) shell-element [7]. While both elements are parabolically curved (Fig. 3) and use standard isoparametric interpolation as a starting point, each departs from the basic recipe in order to properly represent inextensional bending deformation for thin shells.

In particular, the 9.HET element *selectively under-integrates* all stiffness and force terms involving membrane strains to avoid membrane “locking”, and uses a mixture of Lagrange shape functions (for rotations) and Serendipity shape functions (for translations) to avoid “spurious modes” otherwise evoked by reduced integration.

In contrast, the 9.ANS element assumes an appropriate (inextensionally accurate) strain field from the outset, using a modified set of Lagrange shape functions and employing full numerical integration throughout. Due to the fact that the strains are assumed in the generally *non-orthogonal* isoparametric coordinate basis, an apparent advantage of the ANS approach is its decreased sensitivity to element mesh distortion.

Nonlinear Solution Procedures

After performing an *updated-Lagrange* linearization of the equilibrium equations, a modified Newton-Raphson (NR) version of the Riks-Crisfield (RC) arc-length control algorithm [8] is used to trace incrementally the load-displacement curve. The RC procedure was adopted as a convenient means of statically traversing the bifurcation points that arise in shell postbuckling analysis. However, such methods are not foolproof, and special attention by the analyst (*e.g.*, in the selection of imperfections, step-sizes and error tolerances) is often required in the vicinity of closely spaced or *multiple* bifurcation points.

Two kinds of update procedure are required to advance the solution of the discrete nonlinear shell equations from one load-step to the next: (i) a *kinematic* update that “accumulates” incremental *nodal displacement* — both translational and rotational components; and (ii) a *constitutive* update that generates new *element stresses* from the corresponding displacement field.

Only the *rotational* part of the kinematic update is non-trivial.* In the present approach, the rotation increments are used to update *orthogonal triads* defined at each shell node. The triad-update algorithm involves no trigonometric functions, maintains orthogonality at each step and provides a shell-oriented coordinate system in which the *normal rotational* degree of freedom may be eliminated at all shell nodes (except at junctures). Furthermore, once the nodal triads and reference-surface coordinates have been so updated, the *current* element configuration is completely and uniquely defined (see [2] and [9] for details).

The *stress update* is handled via an incrementally objective algorithm [10] that

* Since the translational components of displacement are *vectorial*, translation increments are simply *added* to obtain total displacements and hence update the nodal coordinates.

features a midpoint-rule numerical integration of rate-type constitutive equations. For finite-strain analysis, the constitutive algorithm additionally involves shell *thickness updates* that account for large Poisson effects. These are computed (as in [11]) by recovering the normal strain increments from the constitutive equations via the zero normal stress (ZNS) hypothesis.

Summary of Computational Features

The computational features of the above approach may be summarized as follows:

- Applicable to Both Thin and Moderately Thick Shells
- Rotations May Be Arbitrarily Large
- Strains May Be Large (Except Transverse Shears)
- Resultant Format Yields Cost Savings \propto Number of Layers
- Shell Elements Are Fairly Robust
 - No Locking
 - No Spurious Mechanisms
 - Low Sensitivity to Mesh Distortion (Especially ANS Elements)

Finally, note that while the formulation allows for finite strains and inelastic material behavior, the present analysis simply employs *orthotropic linear-elasticity* within each layer and *does not* account for the material delamination (*i.e.*, damage) observed in the NASA experiment.

§3.2 Implementation (NICE Software Architecture)

The shell-element capabilities mentioned above have been implemented in a modular fashion to facilitate research and transferral to other finite element codes. Some of the specific shell-element functions that are available via independent FORTRAN77 subroutine calls are shown in Figure 4a. A complete description of the shell-element software (including listings) is given in Appendix S of reference [2].

The *host finite-element program* used by the authors is actually a *network* of independently executable programs (or *processors*) which are coordinated via high-level *procedures* written in a mathematically oriented command-language (Fig. 4b). In such an environment, the shell-element software is embedded in a single *processor* and the global solution algorithms are implemented as *procedures*; examples are given in [12].

The software architecture (*i.e.*, utilities) used to construct this particular analysis system is known as NICE (for *Network of Interactive Computational Elements*, [12,13]). Due to the flexibility provided by the NICE architecture and its suitability for nonlinear and coupled-field problems, it is currently being explored by NASA as the basis for a standard generic *testbed* system for Computational Structural Mechanics (see [14] and other presentations therein). One of the motivating factors for developing such a system is the implementational complexity associated with a comprehensive global/local analysis of the present composite-shell problem.

§4. FINITE-ELEMENT MODELS

Due to the physics of the problem, a *full* numerical model of the test specimen is required (Fig. 5). The slight anisotropy emanating from the composite material stacking sequence is only partly responsible for the lack of available symmetry.* As will become apparent, the nonlinear postbuckling response is *inherently* non-symmetric due to the presence of the hole and the participation of many diverse mode shapes.

Several combinations of shell-element type and mesh density were employed during the course of the linear (pre-buckling), stability (buckling eigenvalue) and nonlinear (postbuckling) analyses. Figure 5 shows three representative grids, involving 300, 1500 and 5000 degrees-of-freedom, respectively. These grids correspond to 16, 80 and 256 nine-node elements (or alternatively to 64, 320 and 1024 four-node elements), respectively. Note that there is intrinsic element mesh-distortion in these models — both in-plane and out-of-plane — due to the focus on the hole and the curvature of the shell. However, the elements nearest the hole (where it counts most) have the most regular shapes.

The coarsest grid (1) was used to *verify* the modeling procedure, the finest grid (3) was used exclusively to check convergence of the *linear* and *eigen* solutions, and the intermediate grid (2) became the workhorse for *nonlinear* analysis. Furthermore, as little difference was observed between the 9_HET and 9_ANS elements (§3.1) during the early stages of analysis, the 9_HET element (which is slightly less expensive) is featured in the analytical results that follow.

Boundary conditions were imposed as described in Section 2 and illustrated in Figure 5. To simulate end-shortening, an axial *force* was applied in conjunction with a *degree-of-freedom equivalence* among all axial displacements on the loaded edge. This was done to avoid the use of *specified displacements*, which tend to complicate the adaptive, arc-length-based nonlinear solution algorithm.

* We have confirmed this via numerical experiments with an *isotropic* model.

§5. LINEAR (PRE-BUCKLING) ANALYSIS

Results for the linear pre-buckling analysis are shown in Figure 6. The deformations due to an applied axial compressive load of 22480 lbs (1 kN) are shown magnified by a factor of 10 in the top half of the figure — for grid 2. The corresponding distribution of the axial stress resultant, N_x , along the panel circumference at mid-span is shown in the bottom half of the figure — for both grids 2 and 3.

Regarding the *displacement* solution, convergence of the axial end-shortening, δ , was achieved with grid 2. (Grid 3 yielded less than a 1% increase in end-shortening.) However, the converged end-shortening solution, $\delta = .0316$, is approximately 15% lower than the experimental value; as deduced from the linear portion of the experimental load-displacement curve (Fig. 2). It is presumed that this over-estimation of the axial stiffness is due to the uncertainty in the *nominal* material properties, a conclusion that is reinforced in §8. Thus, to compensate for the mismatch, the lamina principle elastic modulus (E_1) is reduced by a corresponding factor in the subsequent nonlinear analysis (§7).

From a qualitative perspective, the solution shows substantial bending deformation in the vicinity of the hole (Fig. 6a). This suggests that geometric nonlinearity may be important even at relatively low load levels and diminishes the credibility of linear response and buckling-eigenvalue analyses.

Regarding the linear *stress* solution, note that the compressive axial resultant, N_x , is distributed evenly along most of the panel circumference except for a very localized region near the hole. While grid 2 was adequate for the *displacement* solution, grid 3 provides much more accurate resolution of this *stress concentration*. In particular, grid 2 yielded a peak stress concentration factor (SCF) of 2.8; about 14% less than the SCF obtained with grid 3.

As experimental data were not available for verification of the computed stresses, the convergence of the grid 3 stress solution was inferred by comparison with a closed-form (asymptotic) solution due to C.R. Steele (private communication, Stanford University, 1985). While the closed-form solution pertained to a purely *isotropic* panel, the linear finite-element stress solutions for isotropic and quasi-isotropic panels were found to be quite similar. It is also interesting that the SCFs for both the *isotropic* model (3.1) and the *quasi-isotropic* model (3.25) are not very different from the classical SCF for a *flat plate* with a circular hole (3.0).

Finally, note that by strongly *biasing* the mesh towards the hole, it was possible to obtain grid-3 accuracy with grid 2 for the local stress gradients. Such biasing, however, was found to be unnecessary in the subsequent, globally oriented buckling and postbuckling analyses.

§6. STABILITY (BUCKLING) ANALYSIS

Figure 7 shows the first 5 buckling eigenmodes for grid 2. These results represent perturbations about the linear pre-buckling solution described. The eigenvalues, $\lambda = 1.084, 1.106, 1.181, 1.432, 1.582$, are the ratios of the corresponding buckling loads to the axial load applied in the linear pre-buckling analysis. As before, grid 2 seems to provide adequate resolution, with grid 3 giving only a 2% reduction in the first two eigenvalues, and a 4% reduction in the remaining three.

The following observations are important for subsequent computational purposes: (i) the eigenvalues are closely spaced; (ii) the eigenmodes are vastly different in character; (iii) there is no *single* form of symmetry to be exploited computationally; (iv) the first buckling mode is symmetric and bears the most resemblance to the linear pre-buckling solution; (v) the second and third modes possess *skew* symmetries; (vi) the fourth and fifth modes are symmetric; the latter mode featuring practically no distortion of the circular hole; and (vii) higher modes (not shown) look much like those for a cylindrical panel *without* a hole, though the values remain closely spaced.

Finally, it was found that the first (*i.e.*, critical) buckling load is approximately 25% lower than that of an identical cylindrical panel without a hole. Hence, while the influence of the hole on the buckling *load* is only moderate (*i.e.*, relative to the stress concentration factor), its influence on the buckling *modes* is profound. As we shall see, the hole has an even stronger influence on the postbuckling response.

§7. NONLINEAR (POSTBUCKLING) ANALYSIS

In practice, more than just a linearly converged model and an adaptive solution strategy were necessary to obtain a reasonable *nonlinear* solution. One additional pre-requisite was a 15% reduction in the nominal elastic modulus, E_1 (from 19.6×10^6 to 17.1×10^6), to match the linear branch of the load versus end-shortening curve (as explained in §5). Another important ingredient was the specification of *initial imperfections*. In this regard, three versions of the analysis were run: (1) one involving no imperfections; (2) one with an imperfection amplitude of 1% of the thickness applied to each of the first 4 buckling modes (see Fig. 7); and (3) one with an imperfection amplitude of 10% of the thickness applied to each of the first 4 buckling modes. A discussion of the results for these three cases follows.

§7.1 No Imperfections

One would think that the out-of-plane “imperfections” introduced by the linear pre-buckling solution (Fig. 6) would be sufficient to trigger a realistic buckling response. However, this was not the case. With no imperfections, the computed solution path resembled the experiment only up to the descending part of the

load-displacement curve (Fig. 8a). The computed curve then rolled back onto itself with the stiffening branch of the postbuckling curve practically aligned with the *pre-buckling* curve.

To gain further insight, it is useful to look at the deformation and stress histories portrayed in Figure 8b. Shown is a sequence of computational “snapshots” taken at various points (*i.e.*, load steps) on the nonlinear load-displacement curve.* The variation of the axial stress resultant along the mid-span circumference is plotted below each frame.

Note that the deformation starts out (at load step 10) much like the first linear buckling mode (Fig. 7), with inward dimples both fore and aft of the hole, then *articulates* through the second and third modes during the initial postbuckling phase (steps 15–20). This *rotation* of the two dimples about the hole is probably triggered by the bending/twisting coupling inherent in the composite stacking sequence. The dimples continue to rotate and broaden until, at step 40, the pattern begins to resemble the *fifth* linear buckling mode. Evidently, it is this “locking” into mode 5 that is responsible for the excessive secondary stiffening in the load-displacement curve (Fig. 8a). Clearly, mode 5 is an unrealistically *stiff* one, resembling what might occur if a ring *stiffener* had been placed around the hole. This is also evident in the axial stress distribution, where the stress concentration has practically been *shed* by step 40.

§7.2 Large Imperfections

To avoid the unrealistic mode-5 locking observed in the preceding analysis, a fairly large initial imperfection was introduced in the numerical model. This was accomplished by adding an equal measure of each of the first 4 buckling modes to the initial geometry, such that the maximum radial (*i.e.*, shell-normal) displacement in each mode was equal to 10% of the panel thickness. Thus, the magnitude of the combined radial imperfection approached 40% of the thickness at some points. It is emphasized that this rather arbitrary choice of imperfections was designed primarily to minimize the influence of mode 5.

The computed nonlinear response for the imperfect panel is shown in Figure 9. Note that the secondary stiffening branch bears more resemblance to the experiment than did the imperfection-less analysis. Unfortunately, it is also true that the buckling (*i.e.*, peak) load has dropped by about 15%, and now underestimates the *experimental* buckling load by more than 20%. A heuristic explanation is

* Note that the pre-buckling phase of the analysis appears *linear* with respect to the *axial* displacement, δ . However, due to the rapid growth in *radial* displacements (not shown), the analysis is actually quite nonlinear from the outset; which explains the relatively large number of load-steps required on the “linear” branch of Fig. 8.

provided by considering the deformation and stress histories.

In Figure 9b, we see that with the 10%-h imperfections, the buckling pattern leaves mode 1 almost immediately and develops an intensified inward dimple on one side of the panel. Consequently, the full axial load is re-distributed to the *other* side of the panel (see N_x plots in Fig. 9b), accounting for the reduction in both the buckling load and the postbuckling stiffness (fig. 9a).

§7.3 Small Imperfections (the “Bottom Line”)

Finally, the “best of both worlds” was obtained with a 1%-h imperfection in each of the first 4 modes. Compared with the previous analysis, the computed load-displacement curve (Fig. 10) shows both an increase in the maximum load and a decrease in the minimum load, thus bringing the solution more in line with experiment.

The improved performance, obtained by reducing the imperfections, may again be related to the deformation history (Fig. 10b). Here, as in the case without imperfections, two inward dimples develop and proceed to rotate about the hole. Just after buckling, however, one dimple tends to deepen while the other diminishes, and eventually there is a *double* snap-through. This accounts for the double dip in the load-displacement curve (Fig. 10a) and seems to explain why a lower minimum load is obtained with the smaller imperfection.

Still, there are some serious discrepancies between analysis and experiment, namely: (i) a 7% under-estimation of the buckling load, (ii) a 25% over-estimation of the minimum load, (iii) a 30% under-estimation of the postbuckling end-shortening, and (iv) a 10% over-estimation in the postbuckling stiffness. These will be addressed in §9.

§8. CORROBORATION WITH ANOTHER CODE

To support the above results, obtained via the computational procedures described in Section 3, parallel analyses were performed with another finite-element computer code. For this purpose, we employed the STAGS code [15], which has been used for more than a decade by various government agencies and industrial firms (*e.g.*, NASA and Lockheed) to analyze difficult nonlinear shell problems. Another reason for using STAGS is that it features finite-element computational procedures that are substantially different from those used in the present approach, thus adding strength to an analytical comparison.

For the *linear* (pre-buckling and buckling) analysis, excellent agreement was obtained between the STAGS and NICE-based solutions. For completeness, the STAGS runs were performed with two radically different shell-element types, which both converged to the same solution as the NICE/9.HET element, albeit at slower

rates and from *below* in stiffness. (Note that while the axial pre-buckling stiffness of the STAGS elements converged from below, the *buckling loads* converged from *above*).

In particular, the comparison included: (i) the commonly used STAGS/410 element — a flat quadrilateral plate element based on Kirchhoff-Love theory and cubic membrane/bending displacement interpolation; and (ii) the less frequently used STAGS/422 element — a quadrilateral composed of two Kirchhoff-based triangular plate sub-elements with cubic bending interpolation and quadratic membrane interpolation. It is believed that the relatively slow convergence “from below” of the STAGS/410 element is due to *warping sensitivity*, while that of the STAGS/422 element is probably due to the *incompatibility* between membrane and bending displacement fields for non-flat quadrilateral element shapes.*

For the *nonlinear* comparisons with NICE/9.HET, the STAGS/410 element was used exclusively. The resulting load-displacement curves (for 1%-h and 10%-h imperfections) are summarized in Figure 11. The dashed curves represent the NICE/9.HET solutions and the dotted curves represent the STAGS/410 solutions; both were obtained with Grid 2 (Fig. 5).

Note that although the STAGS and NICE-based solutions use different finite-element types, large-rotation update procedures and nonlinear solution strategies, the correlation is remarkable — especially during the postbuckling phase. Even the STAGS zero-imperfection analysis (not shown) resulted in the same excessive postbuckling stiffness as displayed in Figure 8a. One other point: While the STAGS/410 element consistently shows about a 5% higher buckling load than the NICE/9.HET element, thus coming closer to the experimental peak; STAGS/410 is actually *less* accurate — with respect to *discretization* errors — than NICE/9.HET. This follows from the fact that both STAGS/410 and NICE/9.HET converge from *above* in the buckling load. This was confirmed by running the NICE/9.HET element with a *coarser grid*, for which it too showed a 5% higher peak. That non-converged solutions compare better with experiment than converged ones suggests that spatial discretization is not the only source of error here (e.g., see §9).

* It is interesting to note that the STAGS/422 element was used in the related study conducted in [1], where it yielded a 17% more flexible linear solution, and thus agreed better with the linear portion of the experiment. Nevertheless, it has since been found that the boundary conditions were not consistently applied to the element's *mid-side* freedoms in that analysis. By correcting this implementation error, the 17% discrepancy with the other elements has been completely eliminated. Thus, it appears that the “accuracy” obtained in [1] with the STAGS/422 element is due to compensating errors in the nominal material properties.

§9. CONCLUSIONS

§9.1 Summary

The present study may be summarized as follows:

- PURPOSE:
 - Validate continuum-based resultant (CBR) shell formulation
 - Evaluate new shell elements
 - Gain experience with composite-shell postbuckling analysis
- RESULTS:
 - Good agreement in pre-buckling/buckling range
 - Good *qualitative* agreement in postbuckling range
 - Discrepancies due to:
 - *Material properties*
 - *Imperfection sensitivity*
 - *Dynamic effects*
 - *Delamination*

The “good” agreement obtained between the present shell-element formulation and experiment in the pre-buckling and buckling range was possible only after adjusting the nominal material properties so that the linear axial stiffnesses coalesced. The material-property modification was further justified via corroboration with the STAGS finite-element code, which features a substantially different computational approach.

The “best” solution for the nonlinear response was obtained by introducing small (1%-thickness) imperfections corresponding to each of the first four buckling modes. The computed load-displacement curve (Fig. 10a), which again compared well with STAGS (Fig. 11), still showed major discrepancies with the experiment: The discrepancy in the *buckling load* (which is relatively small) may be due to the inadequacy of adjusting only the principal layer elastic modulus, E_1 , rather than the complete set of orthotropic material constants. The discrepancy in the *unloading* phase of postbuckling (*i.e.*, the computed end-shortening *reversal*) is attributed to the quasi-static approximation of what is, in reality, a *dynamic* phenomenon. Finally, the discrepancy in the *stiffening* phase of postbuckling is clearly dominated by damage, *i.e.*, delamination observed in the experiment but not represented in the model.

§9.2 Recommendations

The goal is to eliminate the discrepancies listed with a minimum of computational cost and complexity. To this end, the following steps are recommended:

1) *PERFORM ADDITIONAL EXPERIMENTS*. First, more experimental data are required to verify existing computational capabilities for composite shell post-buckling. For example, panel imperfections should be carefully measured and selected strains and overall deformation patterns should be monitored at frequent intervals. Additionally, an isotropic panel should be tested in order to eliminate the material identification problem and also to provide a standard *benchmark for shell-element evaluation*. The isotropic problem would be valuable for screening out geometrically sensitive elements.

2) *REFINE NONLINEAR GRID*. The nonlinear analysis should be repeated with a finer grid (*e.g.*, Grid 3), as convergence in the linear regime is no guarantee of convergence in the nonlinear regime. Moreover, a study of modal participation both in the imperfections and in the nonlinear response may help establish modeling guidelines for future analysis.

3) *INCLUDE DYNAMIC EFFECTS*. Dynamic effects, which are relatively straightforward to incorporate, should be assessed at the first opportunity. The analysis could be started in a quasi-static mode, switched to an *explicit* transient response algorithm during the unstable phase, and switched again to an *implicit* algorithm during the stable postbuckling phase.

4) *REPRESENT LOCAL FAILURE (DELAMINATION)*. To account for the composite delamination mechanism, appropriate *failure criteria* and *progressive-failure modeling procedures* need to be developed, implemented and evaluated. For shell-based failure criteria, *accurate stress recovery* is essential. Improved stress resolution is required both *along the surface* (*e.g.*, via stress/displacement iterations and adaptive refinement) and *through-the-thickness*, as both normal and transverse-shear stresses can play a dominant role in delamination. Progressive failure may then be simulated by methods ranging from a simplified shell model that *selectively degrades layer properties*, to a full *3D analysis near the hole* with an evolving 2D/3D transition. An intermediate approach is to “split” shell elements along delaminating boundaries. (See Fig. 12.) The simplest approach, however, has obvious implementational advantages.

5) *DEVELOP EFFICIENT GLOBAL/LOCAL ALGORITHMS*. Finally, there is a need to reduce the cost of nonlinear analysis for such problems. The cost of the *global* analysis is dominated by the large number of iterations/steps in the linear-to-postbuckling transition regime. Possible approaches include the reduced-basis

technique [16,17], Thurston's method [18], and improved extrapolators. For the combined *global/local* problem, where material "properties" are changing rapidly during postbuckling, additional features such as line-searches [19], quasi-Newton stiffness updates [20] and nonlinear substructuring may greatly improve efficiency.

We are presently acting on recommendations (2)–(3) and will report the outcome in a forthcoming paper.

ACKNOWLEDGMENTS

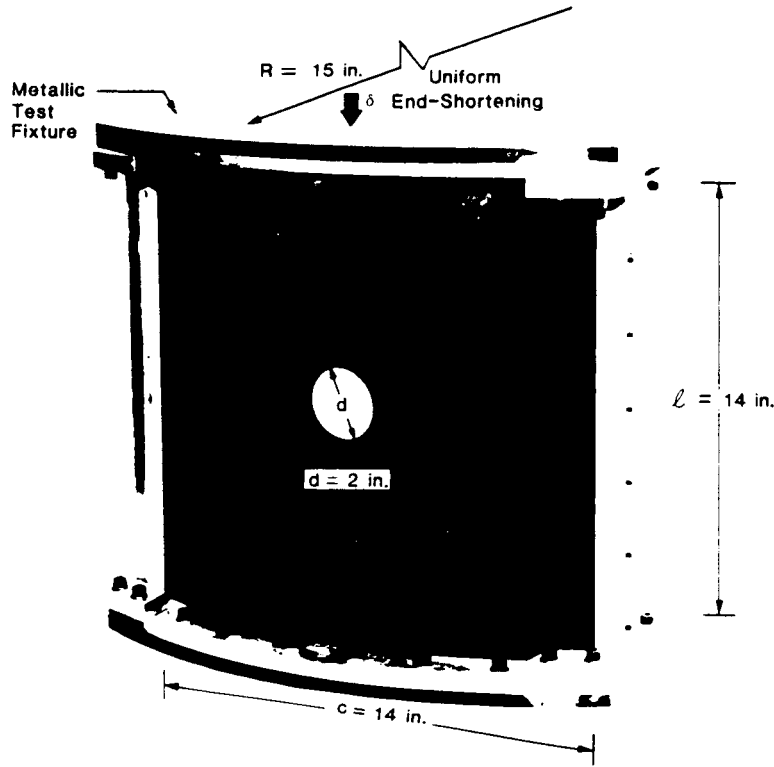
The preparation of this paper was supported by the Independent Research Program of Lockheed Missiles and Space Company, Inc. Ongoing research is being supported by NASA/Langley Research Center as part of their newly launched program in Computational Structural Mechanics.

REFERENCES

- [1] Knight, N.F., Jr. and Starnes, J.H., Jr. (1984), "Postbuckling Behavior of Axially Compressed Graphite-Epoxy Cylindrical Panels with Circular Holes", presented at the 1984 ASME Joint Pressure Vessels and Piping / Applied Mechanics Conference, San Antonio, Texas.
- [2] Stanley, G.M. (1985), *Continuum-Based Shell Elements*, Stanford University Ph.D. thesis, also Lockheed report LMSC-F035839.
- [3] Ahmad, S., Irons, B.M. and Zienkiewicz, O.C. (1970), "Analysis of Thick and Thin Shell Structures by Curved Finite Elements", *International Journal for Numerical Methods in Engineering*, **2**, 419–451.
- [4] Mindlin, R.D. (1951), "Influence of Rotatory Inertia and Shear on Flexural Motions of Isotropic, Elastic Plates", *Journal of Applied Mechanics*, **18**, 31–38.
- [5] Parisch, H. (1981), "Nonlinear Analysis of Shells Using Isoparametric Elements", pp. 47–63 in *Nonlinear Finite Element Analysis of Plates and Shells*, (eds. T.J.R. Hughes et al.), ASME, New York.
- [6] Hughes, T.J.R., and Cohen, M. (1979), "The 'Heterosis' Finite Element for Plate Bending", *Computers and Structures*, **9**, 445–450.
- [7] Park, K.C. and Stanley, G.M. (1986), "A Curved C^0 Shell Element based on Assumed Natural-Coordinate Strains", in *Journal of Applied Mechanics*, June 1986.
- [8] Crisfield, M.A. (1983), "A Fast Incremental/Iterative Solution Procedure that Handles Snap-Through", *Computers and Structures*, **13**, 55–62.
- [9] Rankin, C.C. and Brogan, F.A. (1984), "An Element-Independent Corotational Procedure for the Treatment of Large Rotations", pp. 85–100 in *Collapse Analysis of Structures* (eds. L.H. Sobel and K. Thomas), ASME, New York.

- [10] Hughes, T.J.R. and Winget, J. (1980), "Finite Rotation Effects in Numerical Integration of Rate Constitutive Equations Arising in Large-Deformation Analysis", *International Journal for Numerical Methods in Engineering*, **15**, 1862–1867.
- [11] Hughes, T.J.R. and Carnoy, E. (1981), "Nonlinear Finite Element Shell Formulation Accounting for Large Membrane Strains", pp. 193–208 in *Nonlinear Finite Element Analysis of Plates and Shells* (eds. T.J.R. Hughes et al.), ASME, New York.
- [12] Felippa, C.A. and Stanley, G.M. (1985), "NICE: A Utility Architecture for Computational Mechanics", proceedings of US-Europe Symposium on Finite-Element Methods for Nonlinear Problems, Univ. of Trondheim, Norway (ed. by P. Bergan).
- [13] Felippa, C.A. (1981), "Architecture of a Distributed Analysis Network for Computational Mechanics", *Computers and Structures*, **13**, 405–413.
- [14] Geers, T.L. (1984), "Comments on Proposed Research Program in Computational Structural Mechanics", proceedings of *Workshop on Computational Structural Mechanics*, held 5–7 Sep 1984 at NASA/LaRC, edited by W.J. Stroud.
- [15] Almroth, B.O., Brogan, F.A. and Stanley, G.M. (1979), "Structural Analysis of General Shells, Vol. II: User Instructions for the STAGS(C-1) Computer Code", Lockheed Report LMSC-D33873.
- [16] Noor, A.K. and Peters, J.M. (1983), "Recent Advances in Reduction Methods for Instability Analysis of Structures", *Computers and Structures*, **16**, 67–80.
- [17] Stehlin, P. and Brogan, F.A. (1984), "Analysis of Structural Collapse by the Reduced Basis Technique", pp. 69–84 in *Collapse Analysis of Structures* (eds. L.H. Sobel and K. Thomas), ASME, New York.
- [18] Thurston, G.A., Brogan, F.A. and Stehlin, P. (1985), "Postbuckling Analysis Using a General Purpose Code", presented at AIAA 26th Structures, Structural Dynamics and Materials Conference, Orlando, Florida, AIAA Paper No. 85-0719-CP.
- [19] Crisfield, M.A. (1982), "An Arc-Length Method Including Line Searches and Acceleration", *International Journal for Numerical Methods in Engineering*, **19**, 1269–1289.
- [20] Dennis, J.E. and Moré, J.J. (1977), "Quasi-Newton Methods, Motivation and Theory", *SIAM Review*, **19**, 46–89.

COMPOSITE CYLINDRICAL PANEL WITH CIRCULAR HOLE



NOMINAL COMPOSITE MATERIAL PROPERTIES:

- 16 Layers
- Graphite-Epoxy
- $h_{\text{layer}} = 0.0056 \text{ in.}$
- $h_{\text{total}} = 0.09 \text{ in.}$
- $E_1 = 19.6 \times 10^6 \text{ psi}$
- $E_2 = 1.89 \times 10^6 \text{ psi}$
- $\nu_{21} = 0.38, \nu_{12} = 0.037$
- $G_{12} = 0.93 \times 10^6 \text{ psi}$
- $\{\pm 45/90/0/0/90\}_s$ sym

Figure 1. Knight's problem, NASA test specimen.

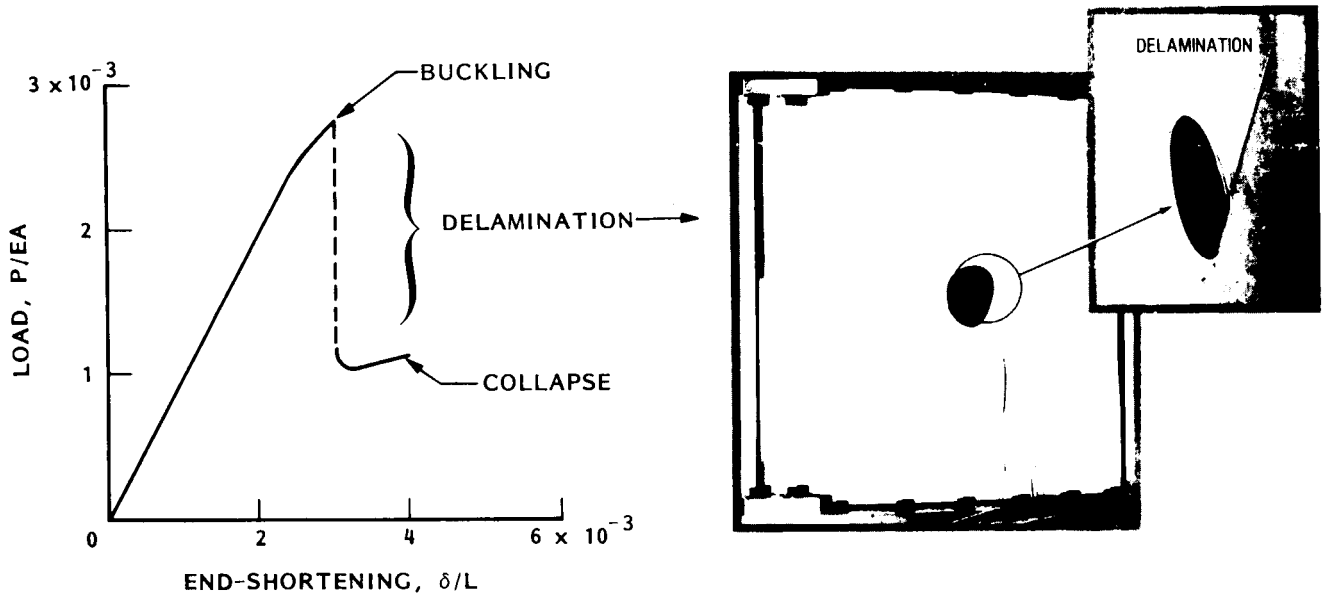


Figure 2. Experimental results.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



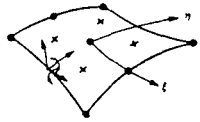

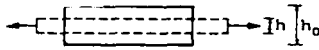
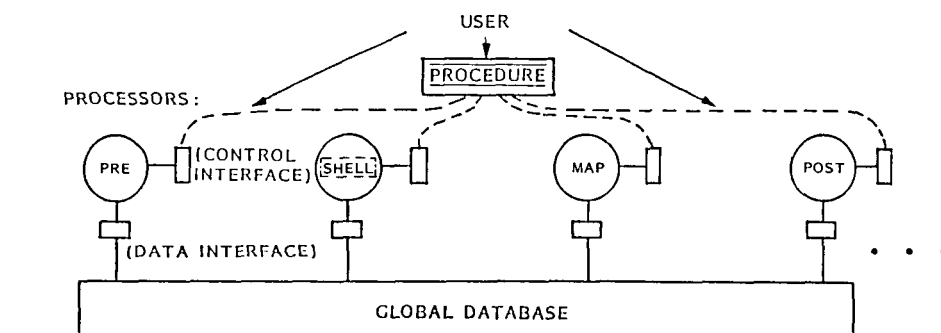
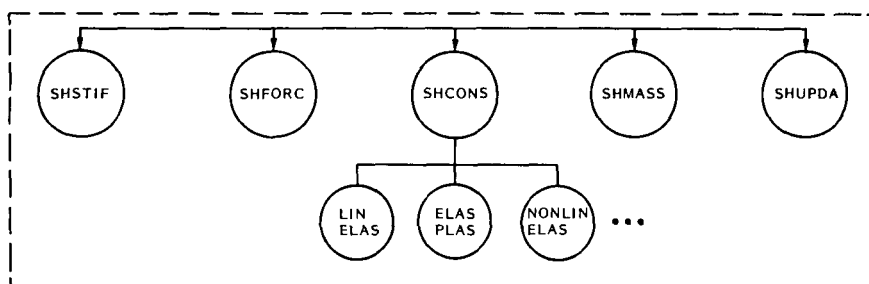
<ul style="list-style-type: none"> ● Continuum-based (CB) shell equations <ul style="list-style-type: none"> - 3-D Continuum equations (equilibrium/constitutive) - Embed shell hypotheses (straight normals, zero normal stress) 	
<ul style="list-style-type: none"> ● Reduce to "resultant" (CBR) form <ul style="list-style-type: none"> - Assume: small transverse-shear strains, mild taper - Preintegrate through-the thickness 	
<ul style="list-style-type: none"> ● Discretize via curved "isoparametric" elements <ul style="list-style-type: none"> - Selective/reduced integration (SRI) - Assumed natural strain (ANS) 	
<ul style="list-style-type: none"> ● Solve nonlinear matrix equations via: <ul style="list-style-type: none"> - Linearization w.r.t. current configuration (UL) - Modified NR algorithm with adaptive (RC) strategy - Nodal triad updates for large rotations 	
<ul style="list-style-type: none"> ● Solve rate constitutive equations via: <ul style="list-style-type: none"> - "Midpoint rule" incremental algorithm - ZNS recovery of normal strains (thickness updates) 	

Figure 3. Computational approach.



a. NICE (Network of Interactive Computational Elements)



b. Shell Element Processor

Figure 4. Implementation.

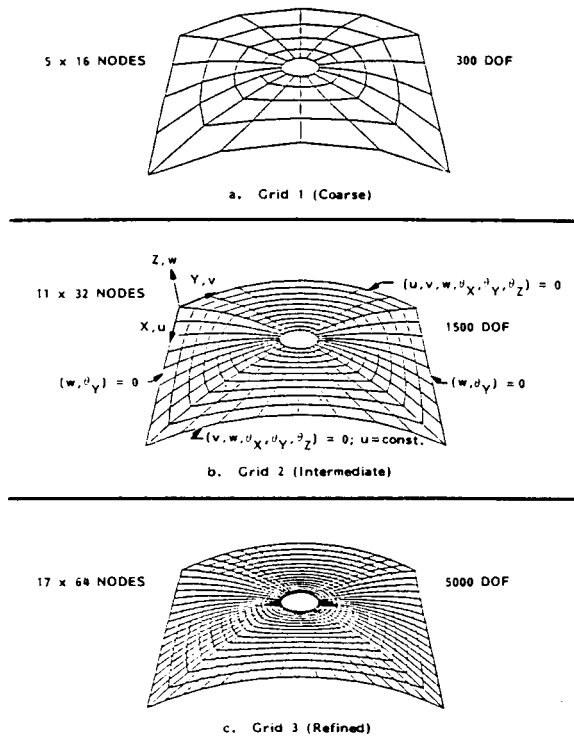


Figure 5. Finite-element models.

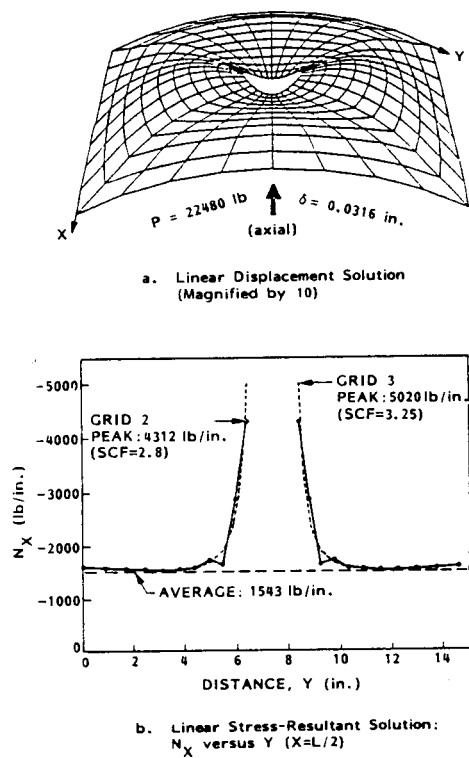


Figure 6. Linear (prebuckling) analysis.

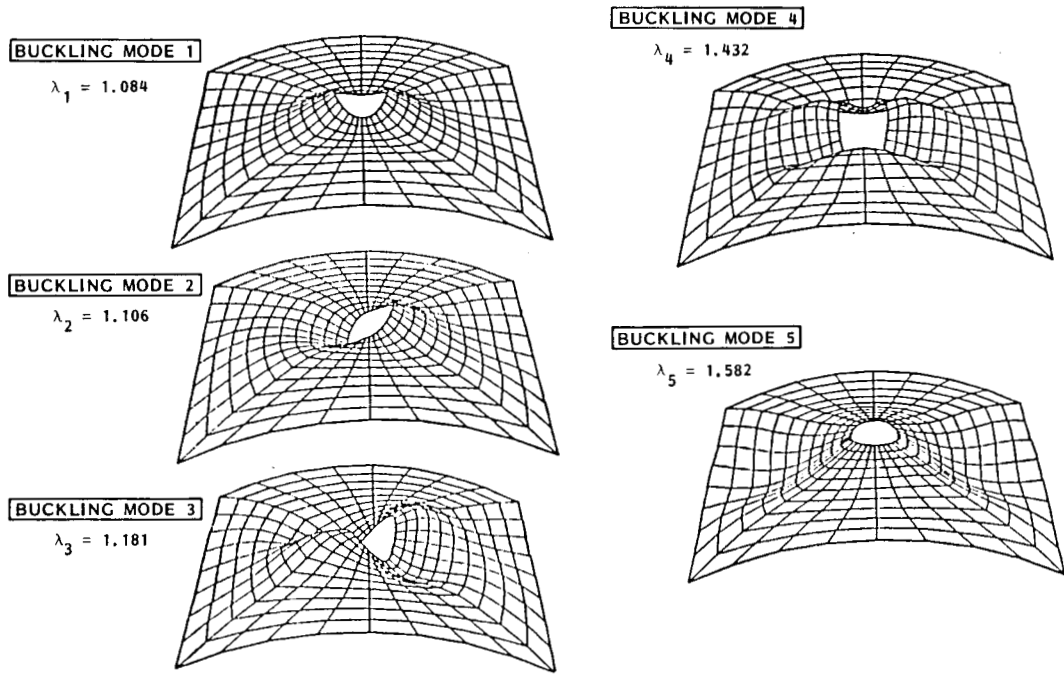


Figure 7. Stability (buckling eigenvalue) analysis.

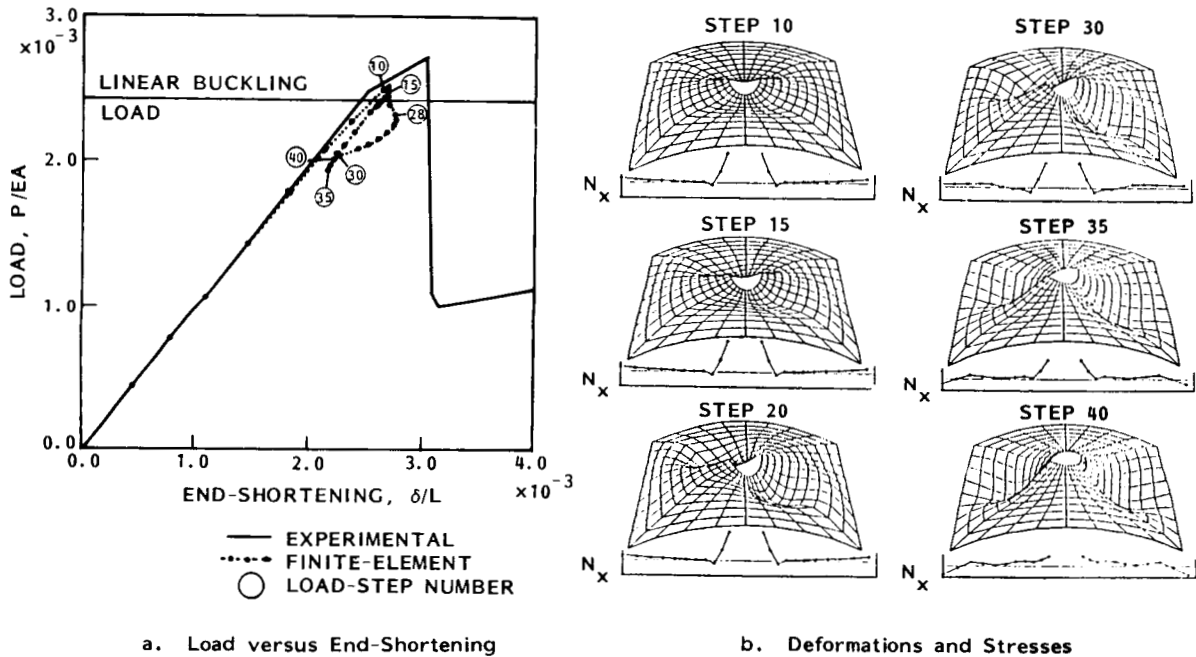


Figure 8. Nonlinear (postbuckling) analysis; no imperfections.

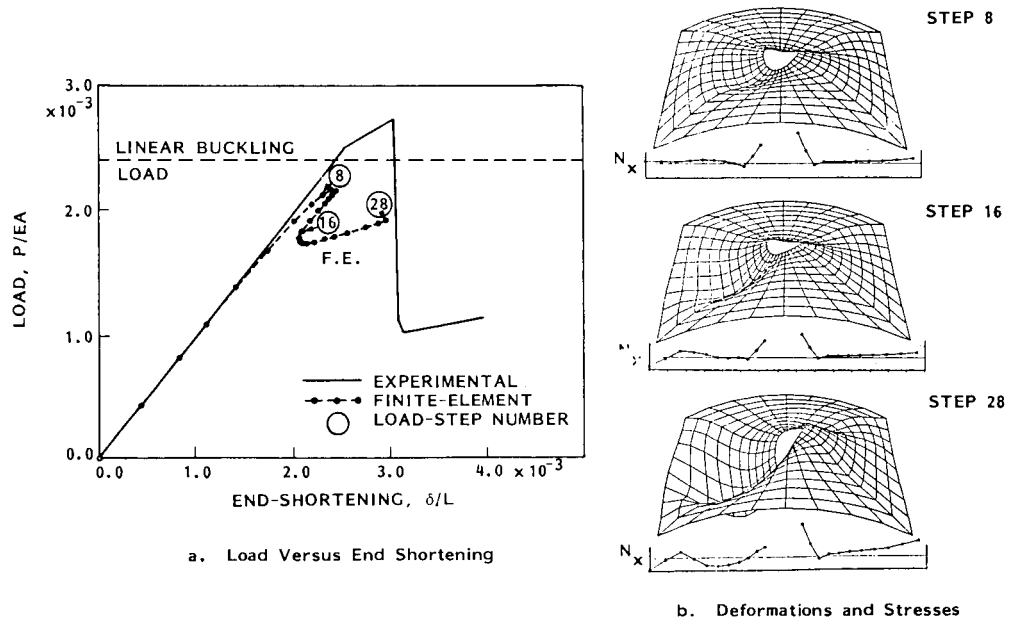


Figure 9. Nonlinear (postbuckling) analysis; 10%-h imperfections.

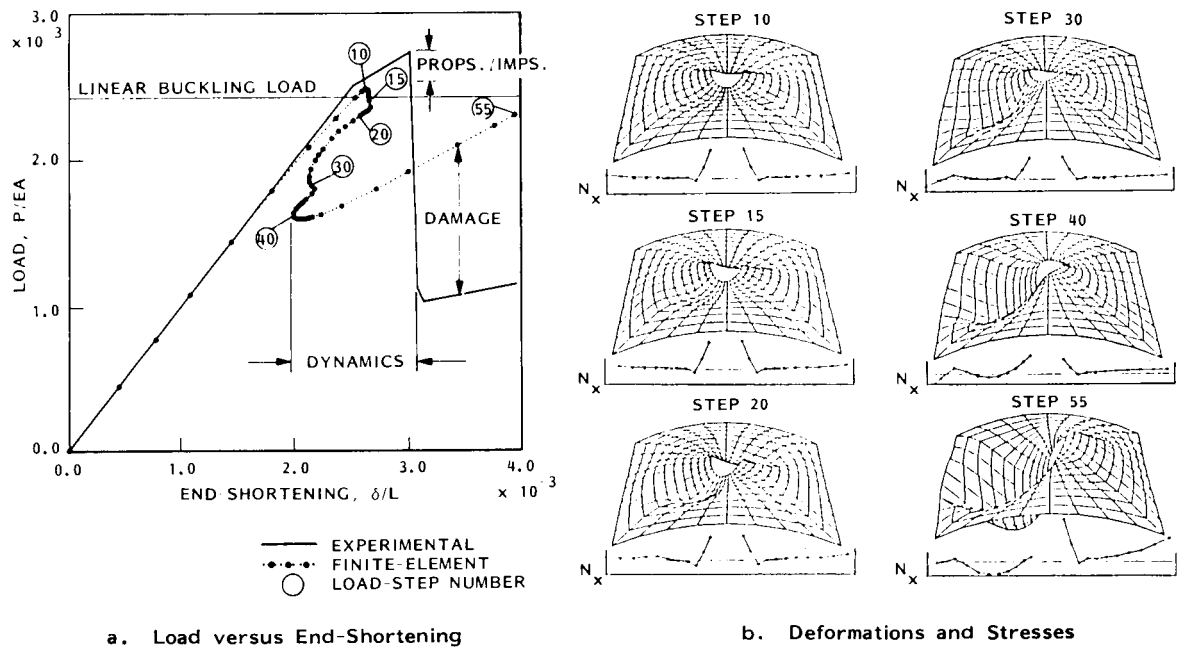


Figure 10. Nonlinear (postbuckling) analysis; 1%-h imperfections.

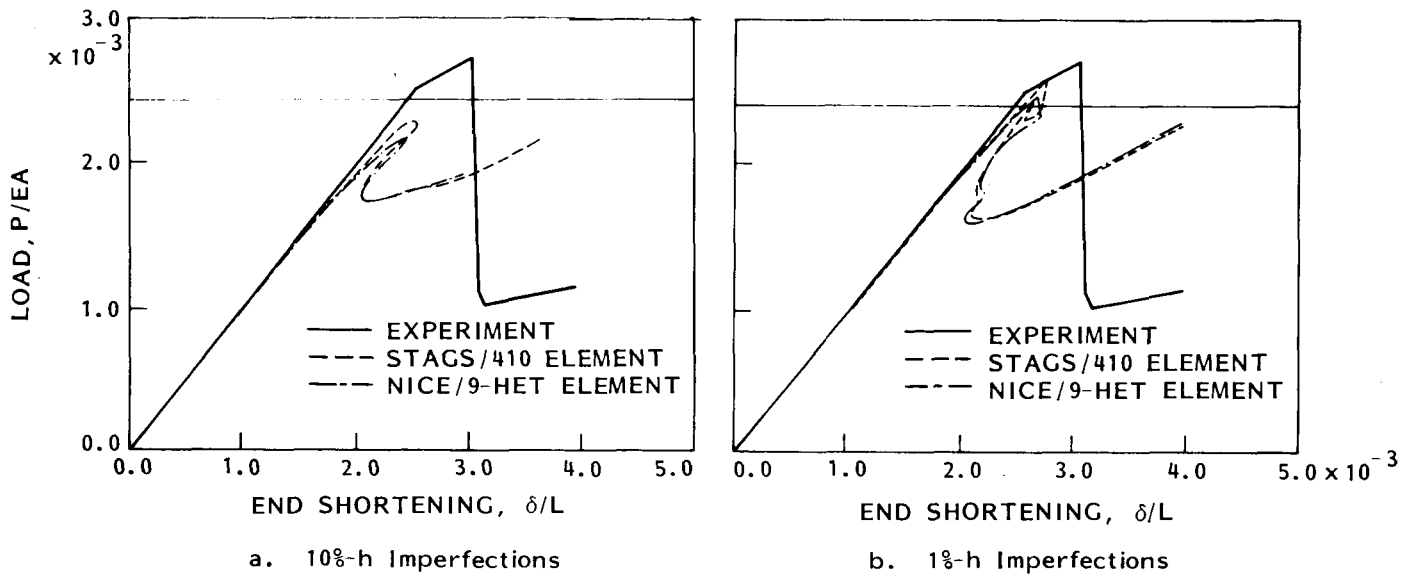


Figure 11. Corroboration with the STAGS code, postbuckling analyses.

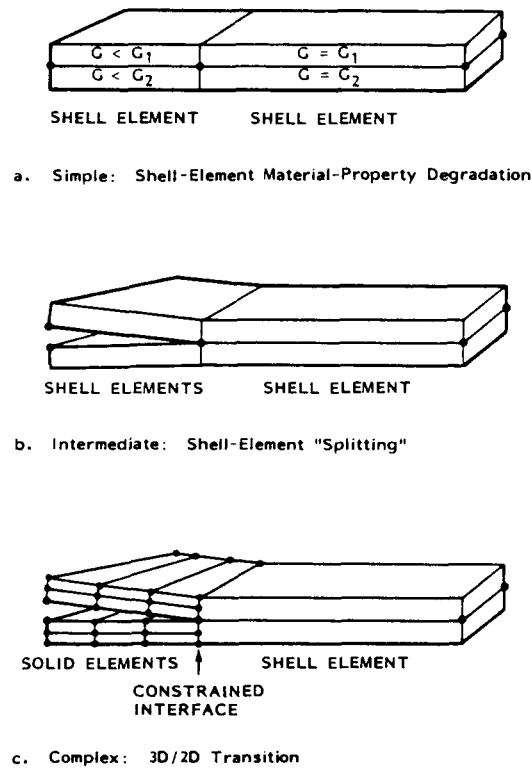


Figure 12. Composite failure-progression models.

A REVIEW OF SOME PROBLEMS IN GLOBAL-LOCAL
STRESS ANALYSIS

Richard B. Nelson
UCLA
Los Angeles, California

INTRODUCTION

The continually increasing power and economy of computers provides the structural engineering and mechanics community with an opportunity and challenge to make major advances in computer intensive areas of analysis, design and nondestructive evaluation of complex structural systems. Certainly the availability of modern computers is making it possible to consider increasingly larger and more complex structural analyses. State-of-the-art commercial grade software is generally available to use for analyzing a variety of linear and nonlinear problems on large mini or mainframe computers. At the same time, structural analysis programs are being "down sized" for use on personal computers.

Yet, given all of these advances, some important and perhaps even critical problems are developing which must be resolved if the remarkable improvements in computer-based analysis and design over the past 10 years are to continue and the structural engineering community is to take full advantage of the new computing power. Several problems should be briefly mentioned.

First, modern structural analysis software is generally a proprietary product of an active and very competitive commercial software industry. As such the software is beyond the control of the engineers who are almost completely dependent on it for performing structural analysis and design. Thus, they do not directly control the computer analysis and therefore are not able to fully understand the results they obtain. As a consequence engineers must rely on faith and earlier experience with given software to justify their analyses and subsequent designs.

Second, the software packages available to the engineering community are enormously complex, so that even if source listing of the programs were available, few engineers would be able, much less willing, to learn how the program works. Thus, the sheer size and complexity of the software encourages both the user and the software vendor to let well enough alone. Incremental changes in software are surprisingly difficult and most software requires almost continuous support by a technical staff.

As a result little incentive exists for either engineers or software firms to push for major modifications in existing software or to develop fundamentally new and more powerful software. Unfortunately, in addition to being hampered by size and complexity, today's software is a product developed for yesterday's computers. For example, most programs are written using logic designed around one-dimensional arrays to store compacted stiffnesses and column solution techniques to solve equations in order to minimize storage requirements. However, modern computers have almost unlimited (virtual)

memory and array processors which would be more effectively used if the software had a different program architecture. Thus, much of today's structural analysis software is unable to take full advantage of the most advanced computing machinery. A major revision of an existing program is a tremendous undertaking and, if developed, may have significant short-term costs to both the software developer and the user.

Finally, it should be recognized that a number of "research" areas in structural mechanics have reached the development stage and are starting to be used by the structural engineering profession. Examples are structural life predictions using fracture mechanics, structural system identification, structural design optimization, semi-automated, adaptive computer modeling, and the analysis of infinite domain problems. The development of these areas has been and will continue to be slowed until the time when the necessary analysis techniques become generally available in commercial grade programs.

If new generation software is to have maximum impact on the structural engineering profession it must be configured so that basic analysis can be used as a tool in a more general comprehensive engineering program. The software also should be as adaptable as possible to the evolution of computing machinery.

It is the purpose of this presentation to provide some areas of structural engineering which are not well served by today's software and which should be given serious attention by developers of future structural analysis programs. In keeping with the theme of this workshop session, several aspects of global-local stress analysis will be discussed, with attention drawn to both the nature of the problem and the type of computational software which should be developed to investigate the problem.

BASELINE MODELING CONCEPTS

Perhaps the most difficult decision the engineer analyst must make is the choice of the proper mathematical model for use in investigating structural behavior (fig. 1). Generally the analyst must choose first the dimensional level of the model, e.g., 1-D (truss/beam) technical theory, 2-D technical theory, 2-D continuum modeling, and 3-D continuum modeling or a model composed of a mixture of some or all of the above. A major consideration in this choice is the available computer element library (which, for most programs is reasonably complete) and the performance of given finite elements. Also, elements and/or procedures for interfacing different types of element types are very important for problems where mixed element types are to be used.

The analyst must also specify the level of physics to be treated by the computer program, especially the material constituency, the effect of initial stress, large motion (stability), dynamical response and nonlinear effects. The analyst should be free to investigate all these effects if necessary and not be constrained by limits on the features of a given software package (e.g., lack of a geometric stiffness matrix for a 3-D continuum finite element). As may be shown with a number of systems, structural behavior may initially be linear, elastic and even quasistatic in nature under given loads, but given "small" changes in configuration, such as a hole or notch in a critical section, the problem may be fundamentally different involving

nonlinear elastoplastic response, large deformation and leading to large-scale failure.

Thus to investigate a system thoroughly the analyst needs to have access to a very complete analysis package involving comprehensive physics and numerical modeling.

The effective use of modern finite-element software depends on the basic skill of the analyst/engineer, who may be tempted to replace insight and basic knowledge of structural analysis with degrees of freedom (DOF's). Such analyses first of all cost more than is necessary. To make matters worse large DOF models with excellent graphical characteristics may contain low resolution physics. (However, three dimensional models may be much easier to present to clients or company executives!)

Consider the cantilever beam shown in figure 2. The simple 2 DOF model gives excellent beam deflections from which the trained engineer can obtain a wealth of accurate stress data. The planar 96 DOF model is in many respects less accurate. For example, the (exact) cubic lateral displacement is only approximately modeled by a series of parabolas. The stresses near the tip and near the left end are captured with little, if any, additional precision. The 192 DOF planar model is, if anything, less precise than the 96 DOF model since lower precision 4 node elements are being used. Finally, the complex 3-D model with 768 DOF is still less accurate since the plane stress assumption is no longer built in to the model and now must be obtained (approximately) through the solution of a large system of equations.

This example shows the efficiency that can be achieved using a well-thought-out finite-element model but it also shows some difficulties that can result. The deceptive ease with which software can be used invites abuse by unwary or poorly trained users. It is the author's opinion that some (optional) diagnostics should be available to warn or guide users in the generation of finite-element models.

One of the most common uses of finite-element analysis is to investigate the behavior of structures in the vicinity of sudden variations in structural geometry or configuration in which singularities in the stress field or stress concentrations may occur. This area of analysis will be termed near-field modeling (fig. 3). Two different approaches for investigating the behavior of structures in the vicinity of stress concentrations reflect basic concepts global-local stress analysis. In the first approach, the finite-element model of the general structure away from the area of stress concentration (a hole in fig. 3) is coupled with an analytical solution [1]. The finite-element model may or may not extend into the region $R < R_L$ influenced by the analytical solution. For the problem shown the analytic stress field is in the form of Kirsh's solution [2] with an unknown stress factor $\bar{\sigma} \neq \sigma_0$. The interaction between the exterior finite-element model and the interior analytical model is used to find $\bar{\sigma}$ as well as the nodal displacements and internal stresses in the exterior grid.

The approach requires an analytical solution simple enough to be effectively used in a computer analysis. This approach appears to be limited to primarily isotropic homogeneous elastic structures with very simple stress concentrations, and for simple loadings.

Current software packages generally do not contain a library of analytical solutions and, if the numerical and analytical regions overlap, may not have the capability for generating and solving the necessary equations of equilibrium.

A different approach to solving the same problem is to simply model the entire structure with a finite-element model using a mesh which is sufficiently finely zoned near the hole to capture the stress concentration (fig. 4). The generation of such a mesh requires a basic understanding of the physics of the stress concentration, especially the characteristic lengths of the decay of the stress concentration, and also a good understanding of the capabilities of the finite elements chosen to model the structure.

An assessment of the quality of the finite-element model may be difficult if the analyst has a limited understanding of the finite elements being used to represent the physics, i.e., a problem if the analyst is using a program as a "black box."

In order to check for a valid solution the engineer often uses a finer mesh to re-analyze the problem especially in the vicinity of the apparent singularity indicated from the previous solution. This "field" application of the patch test is commonly used to check the convergence characteristics of a computer simulation. It would be helpful if it were necessary to generate only the data in the revised part of the structure and the data generated for the unchanged part could be reused. This is a simple task and yet one which is not commonly available in commercial computer programs. This feature may become quite important if the physics is more complicated than first believed, such as might be the case if a central stiffener were present, which would force the analyst to consider the stress concentration due to the hole and the nearby shear lag problem due to the interrupted stiffener.

The problem of analyzing systems in which a part of the system involves an infinite or semi-infinite continuum is a perplexing and difficult problem, since an unlimited number of finite elements may be needed to model the complete system. The idea of using an analytical solution for far-field behavior together with a finite-element model of the near-field structure is a compelling one and has been used by a number of authors [3,4]. In figure 5, R_M denotes the outer radius of the finite-element model.

As in the near-field problem, an analytical solution that can be effectively utilized in the context of a finite-element analysis is required. The need for using an analytical solution has restricted the method to problems involving elastic isotropic homogeneous media and to a relatively small class of static or forced vibration problems. Recently efforts have been undertaken to generalize the method to problems where the medium was orthotropic or layered using finite-element solutions for the far-field response of layered media in place of analytical functions.

In this approach, analytical solutions may have a number of terms, each with some characteristic factor F_i which must be related to the applied load. As in the near-field case, the region of the analytical solution may or may not extend into the region of the finite-element model.

The technique is not easily applied using conventional finite-element analysis packages since neither the definition of the far-field solution nor the techniques for matching the finite-element grid with the far-field analytical solutions are contained in the programs.

The far-field analysis problem is often investigated by using a so-called media island to treat structure media interaction (fig. 6). The finite-element model typically extends as far as is economically practical from the site of interest. On the outer boundary, some special procedure is used to make the boundary a transmitting one, i.e., to permit outgoing waves to pass out of the computational grid and eliminate spurious reflections that might contaminate the solution. Typical boundary treatments are to use dampers, or special paraxial boundary elements [5], or recently, to use a so-called boundary zone superposition zone [6,7] to trap and cancel spurious waves.

The latter method appears quite promising and requires only some basic knowledge of wave speeds in the boundary zone. The region in the interior may behave in a linear or nonlinear fashion. An important characteristic of the boundary zone superposition method is that it is very simple to program and may be used in principle with any finite-element software package. Unfortunately, in practice this is not the case since the analyst may be using "black box" software over which he has no control.

The development of a practical tool for near-field/far-field analysis has major implications for such problems as, using ultrasonics for nondestructive testing, developing sensor/control systems on very large spacecraft, and studying impact of large bodies, such as spacecraft and Shuttle-type transport vehicles.

The behavior of connections in structures is a persistent problem for structural engineers, especially on structures such as a space station which may have hundreds or even thousands of connections. Unlike terrestrial systems the connections on spacecraft may be very lightly loaded and therefore play a very important role in determining structural response of the overall system. The connection is a physical stress concentration (or substructure) in which structural characteristics may be quite complex (fig. 7). Yet for purposes of analyzing a very large system the highly resolved behavior of a connection must be consistently and appropriately reduced to a level usable for the analysis of the large-scale system.

The development of a simple substructure model which gives the essential behavior of the substructure in a global system is a non-trivial task. Certainly "downsizing" a highly resolved model of a connection to a much simpler model suitable for use in a global analysis should be based on a consistent formulation in which overall internal energy under specific deformation patterns is maintained, and the model should account for appropriate rigid-body behavior.

The use of connection elements has been incorporated in a number of finite-element programs, especially for use in piping analyses in the nuclear power industry. However, the software for an analysis such as described for the connection will have to have the capability of giving the user control over the definition of the element (as compared to having access to one of several basic connection elements in a general element library).

A different type of substructure problem which is encountered in global-local finite-element analysis is the case where the important physics is in a small region of a large structure (fig. 8). This is the typical case in a problem involving the analysis of crack growth using fracture mechanics. In order to determine the rate at which the crack shown on the structure will grow given applied cyclic loadings, the stress intensity factor at the front of the crack must be determined for the crack as it grows during loading, or alternatively, the strain energy release rate. This is accomplished by analyzing the structure with a given length of crack, and then releasing the connection between elements at the tip of the crack (allowing it to advance for one element) and reanalyzing the structure.

This procedure amounts to a model revision; thus, the entire structural analysis problem must be reestablished and resolved. This is only practical on a large system involving many thousands of DOF's if the surrounding structure is treated as a substructure and the crack growth region as the primary structure (which may be repeatedly modified to perform the strain energy release rate calculation). This type of analysis can be done using available commercial software, but only in a one-solution-at-a-time mode. It would be very helpful if the procedure could be carried forward in a semiautomatic manner that would require substantial software development.

This analysis is very important in making safe-life predictions for critical components in aircraft and spacecraft. Of course, the problem is much more complicated if the direction of crack growth is unknown, since the finite-element models of the substructure and structure could not be determined prior to analysis. In short the finite-element model would have to be adaptive.

Based on the comments in figure 8 it is evident that finite-element modeling must be adaptive in order to make safe-life predictions, a process which now involves the engineer analyst directly. In fact, considerable research has been done to develop semi-automatic, adaptive finite-element mesh generators [8-10]. These procedures operate in basically one of two ways, refinement of the mesh itself, using similar finite elements the same order of approximation within each element (H-convergence), or leaving the grid fixed but refining the physics within each element (P-convergence). Different strategies are used to assess the quality of a solution for a given finite-element grid. The same information is then used to revise the model and improve the solution.

In response to a test problem proposed by NASA as a vehicle for discussion at the workshop (fig. 9) a simple highly idealized model of the structure was prepared by Dr. Paolo Roberti and analyzed using his algorithm [10]. This algorithm uses triangle constant strain finite elements and H-convergence. The results are remarkable, giving almost a map of the stress concentration in the vicinity of the hole in the stiffened panel, figure 10a-f. Of course, this analysis was conducted only for a linear static solution. In reality, the presence of the hole in the panel may lead to instability or even failure. Nevertheless, the analysis is important in developing a finite-element model with a specified precision.

Approaches such as this only hint at the tremendous problem solving power that can be brought to bear on structural engineering problems if the software

available can be designed to be flexible enough to adopt new and different concepts in analysis.

CONCLUDING REMARKS

The various types of local-global finite-element problems point out the need to develop a new generation of software. First, this new software needs to have a complete analysis capability, encompassing linear and nonlinear analysis of 1-, 2-, and 3-dimensional finite-element models, as well as mixed dimensional models. The software must be capable of treating static and dynamic (vibration and transient response) problems, including the stability effects of initial stress, and the software should be able to treat both elastic and elasto-plastic materials.

The software should carry a set of optional diagnostics to assist the program user during model generation in order to help avoid obvious structural modeling errors. In addition, the program software should be well documented so the user has a complete technical reference for each type of element contained in the program library, including information on such topics as the type of numerical integration, use of underintegration, and inclusion of incompatible modes, etc. Some packaged information should also be available to assist the user in building mixed-dimensional models.

An important advancement in finite-element software should be in the development of program modularity, so that the user can select from a menu various basic operations in matrix structural analysis, including matrix formulation and storage, assembly (by row or column), solution (by row, column or wave front), and method of time integration. Most important, the software should permit the user/analyst to link to the computer program his own specialized software. User programs might include formulation of (substructure) stiffness matrices, specialized solution packages (matrix inversion, partial inversion), time integration and, for nonlinear problems, input of different types of materials.

The next generation of finite-element software also should be developed with the idea of analysis serving as a basic tool in design, system identification and optimization.

The implementation of adaptive finite-element modeling techniques in commercial grade software will have a major impact on the structural engineering community which now invests a significant effort on basic analysis, especially in the modeling, solution, and remodeling cycle. A number of problems in nonlinear structural analysis will also benefit from adaptive computer modeling, such as making safe-life predictions for structures using fracture mechanics concepts.

Hopefully, a new generation of software can be developed with many, if not all, of the features described. If it is possible to do so, then structural analysis software will become a much more complete, versatile and reliable tool for the structural engineer.

REFERENCES

1. Bradford, L. G., Dong, S. B., Nicol, D. A. C., and Westmann, R. A., "Application of Global-Local Finite Element Method to Fracture Mechanics," Electric Power Research Institute Report #EPRI NP-239 (Research Project 299-1) Technical Report 1, September 1976.
2. Timoshenko, S. P., and Goodier, J. N., Theory of Elasticity, Third Edition, McGraw Hill, New York, 1970.
3. Muki, R., and Dong, S. B., "Some Remarks on the Use of Asymptotic Solutions in Global Local Finite Element Analysis for an Elastic Half Space," Recent Research on Mechanical Behavior of Solids, University of Tokyo Press, 1979, pp. 55-78.
4. Medina, F., and Taylor, R. L., "Finite Element Techniques for Problems of Unbounded Domains," International Journal for Numerical Methods in Engineering, Vol. 19, No. 8, August 1983, pp. 1209-1226.
5. Cohen, M., and Jennings, P. C., "Silent Boundary Methods for Analysis," Computational Methods for Infinite Domain Media-Structure Interaction, ASME, 1982, pp. 183-204.
6. Kumar, R. R., and Marti, J., "A Non-Reflecting Boundary for Explicit Calculations," Computational Methods for Infinite Domain Media - Structure Interaction, ASME, 1981, pp. 183-204.
7. Muki, Y., "A Radiating Boundary Method for Linear and Nonlinear Transient Finite Element Analysis," M.S. Thesis, UCLA School of Engineering and Applied Science, 1985.
8. de S. R. Gago, Kelly, D. W., Zienkiewicz, O. C., and Babuska, I., "A Posteriori Error Analysis and Adaptive Processes in the Finite Element Method: Part I - Error Analysis," International Journal for Numerical Methods in Engineering, Vol. 19, No. 8, August 1983, pp. 1593-1620.
9. de S. R. Gago, Kelly, D. W., Zienkiewicz, O. C., and Babuska, I., "A Posteriori Error Analysis and Adaptive Processes in the Finite Element Method: Part II - Adaptive Mesh Refinement," International Journal for Numerical Methods in Engineering, Vol. 19, No. 8, August 1983, pp. 1621-1656.
10. Roberti, P., "Self Adaptive Mesh in Stress Analysis," Ph.D. Dissertation, UCLA School of Engineering and Applied Science, 1984.

BASELINE MODELING CONCEPTS

- Dimensional Level of Model

- Technical Theory
 - 1-D Truss/Beam
 - 2-D Panel/Plate/Shell
- Continuum Representation
 - 2-D Plane Stress, Plane Strain
 - nth order Symmetry
 - 3-D

- Physical Requirements of Model

- Material (Elastic, Plastic, Anisotropic)
- Initial Stress (Stability)
- Dynamics (Vibration, Transient Analysis)
- Nonlinear (Large Deformation, Separation)

Figure 1

FINITE-ELEMENT MODELING

- Which Model is Better?

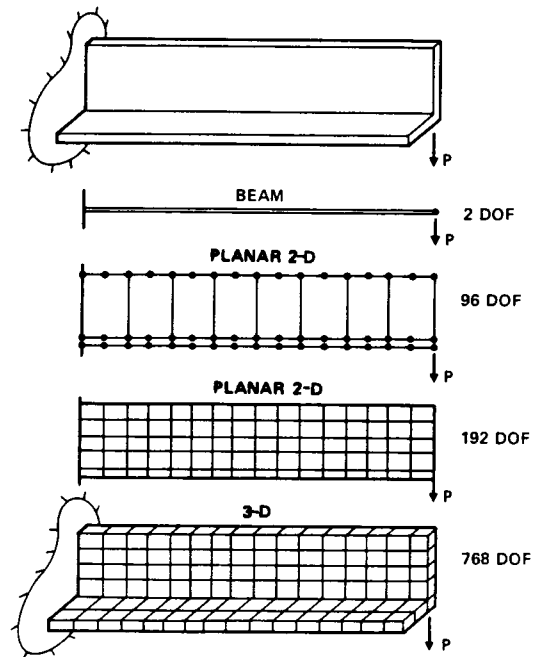


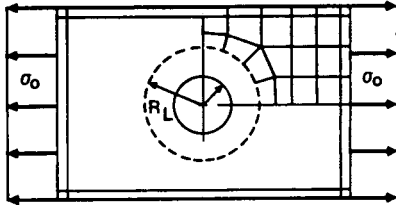
Figure 2

NEAR-FIELD MODELING

Singularities/Stress Concentrations

- Finite-Element Model with Near-Field

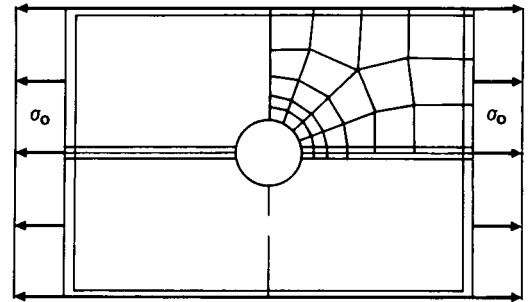
Analytical Solution



- Analytical Solution(s) Required
 - Capable of Being Evaluated and Utilized
- Restricts Approach to
 - Isotropic, Homogeneous Elastic Systems,
 - Very Simple Regions, Simple Loadings,
 - Static Problems, or Forced Vibrations

Figure 3

- Finite-Element Modeling with Mesh Refinement



Understanding of Characteristic Lengths of Physical Processes Required

Evaluation of Model, Solution Difficult

Model Refinement Capability Important

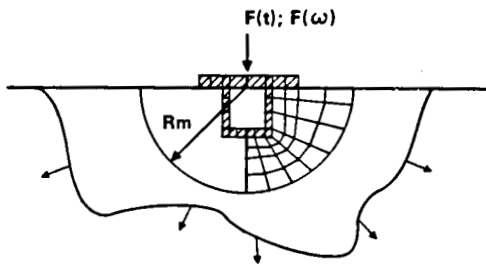
Figure 4

FAR-FIELD MODELING

Finite Structure Imbedded in Infinite Continuum

- Finite Element Model with Far Field

Analytical Solution



Analytical Solution(s) Required

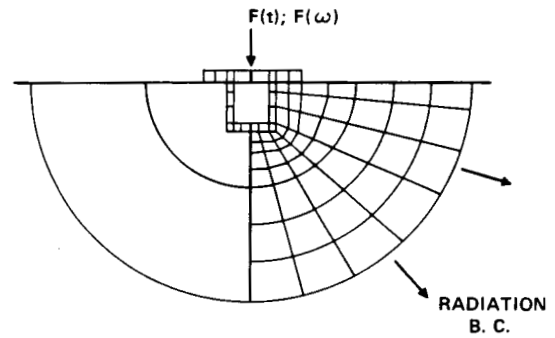
Capable of Being Evaluated and Utilized

Restricted to Isotropic Homogeneous Elastic

Far Field, Static or Forced Vibration

Figure 5

-Finite-Element Model In Continuum Island



Formulation Requires Special Boundary

Conditions to Insure Radiation

Dampers

Reflecting Zones

Knowledge of Wave Transmitting

Characteristics of Medium Required

Figure 6

SUBSTRUCTURE CONCEPTS

Use of Highly Resolved Models

- Imbedded Substructures

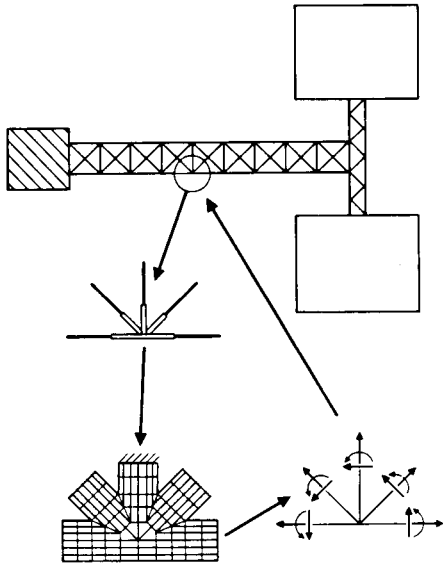


Figure 7

- Surrounding Substructure

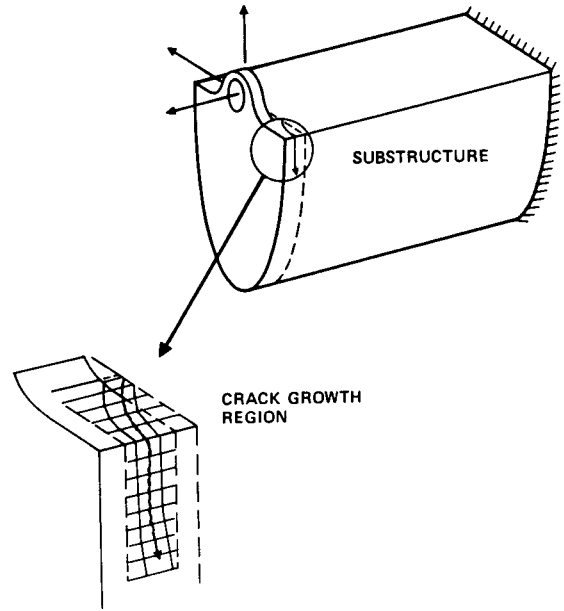


Figure 8

ADAPTIVE FINITE ELEMENT MODELING

BLADE-STIFFENED PANEL WITH DISCONTINUOUS STIFFENER

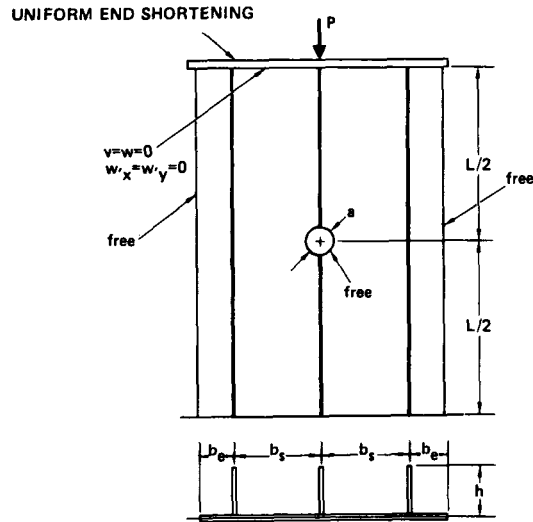


Figure 9

RESULTS OF ROBERTI'S ALGORITHM FOR SUCCESSIVELY REFINED MESHES TO CAPTURE STRESS CONCENTRATIONS IN NASA TEST PROBLEM

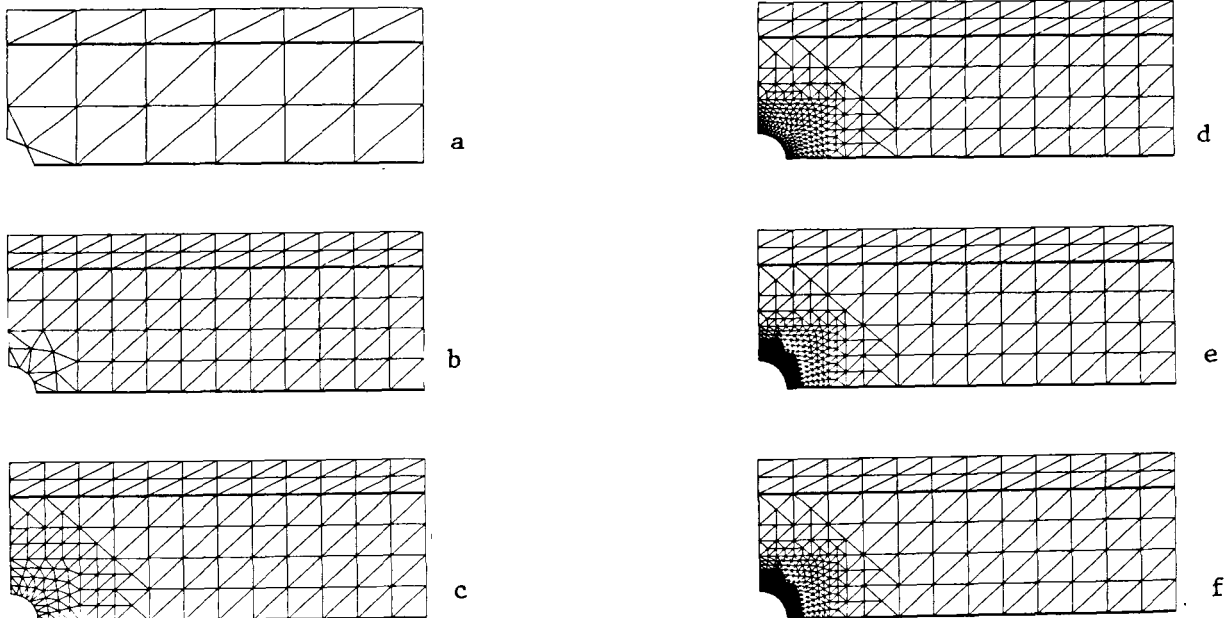


Figure 10

SOME COMMENTS ON GLOBAL-LOCAL ANALYSES

Satya N. Atluri
Georgia Institute of Technology
Atlanta, Georgia

ABSTRACT

The main theme of this paper concerns methods that may be classified as global (approximate) and local (exact). Some specific applications of these methods are found in:

- (1) Fracture and fatigue analysis of structures with 3-D surface flaws
- (2) Large-deformation, post-buckling analysis of large space trusses and space frames, and their control
- (3) Stresses around holes in composite laminates

A typical engineering problem is illustrated in figure 1, which shows a corner flaw at the intersection of a nozzle and a pressure vessel. The shape of the surface flaw may often be approximated mathematically as quarter-elliptical or quarter-circular. For the problem shown in figure 1, wherein the crack is located in the longitudinal plane of symmetry of the structure, only the so-called Mode I conditions prevail. In figure 1, the presence of a traction-free crack, in an otherwise unflawed solid, alters the stress-state only locally. From a viewpoint of fracture mechanics, however, the main quantities of interest are only the stress-intensity factors (strengths of asymptotic stress singularities) near the crack front. For analyzing fatigue crack growth and crack instability under thermal shock various flaw sizes and shapes need to be considered. The primary objective of analysis is to determine the variation of the Mode I stress-intensity factor along the border on the surface flaw.

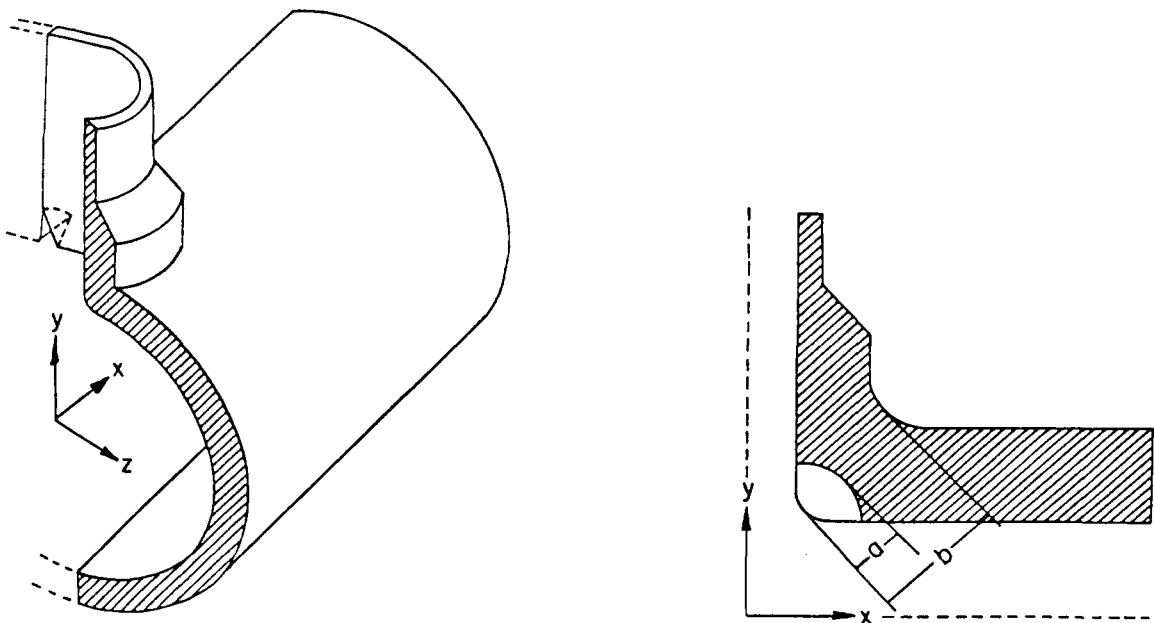


Figure 1. Corner surface-flaw at the pressure-vessel-nozzle intersection.

Figure 2 shows the schematic of a 12-bay space frame. The equations of dynamic motion of the frame, assuming large deformations and plasticity, may be written as:

$$\tilde{M}^{(N+1)} \ddot{\tilde{X}} + \tilde{C}^{(N+1)} \dot{\tilde{X}} + {}^t\tilde{K} \Delta\tilde{X} = \tilde{f}_C + Q_E - {}^{(N)}\tilde{R}$$

where \tilde{M} is the mass matrix, \tilde{C} the matrix of passive damping, ${}^t\tilde{K}$ the tangent stiffness matrix (which includes the effect of large deformation and plasticity), \tilde{f}_C the control-actuator force, Q_E the external load, ${}^{(N+1)}\ddot{\tilde{X}}$ the acceleration vector at time t_{N+1} , ${}^{(N+1)}\dot{\tilde{X}}$ the velocity vector at t_{N+1} , $\Delta\tilde{X}$ the incremental displacement between t_N and t_{N+1} , and ${}^{(N)}\tilde{R}$ the internal-force vector at t_N . In order to implement the control algorithms in an efficient manner, the order of the above system of equations must be as small as possible (i.e., each frame member must be modeled by no more than one finite element). Further, the control must be implemented for pulse-type loading of high intensity, such that the above system of equations must be integrated directly rather than using a modal-decomposition. Also requirements of on-line control may necessitate that ${}^t\tilde{K}$, \tilde{C} , and \tilde{M} be known explicitly (in closed form) for arbitrary values of deformation, without the need for introducing approximate shape functions for deformation of each element and without the need for any numerical integrations over each element. In figure 2, the object of inquiry is what effect does local (member) instability have on global (system) stability? How can we control the dynamic deformations locally to improve global behavior? Each member may be treated as a truss member, or a 3-D beam-type member, depending on joint design. How can local effects be accounted for simply and efficiently, so that algorithms for control of dynamic motion may be implemented, on line, using on-board computers in a large space structure?

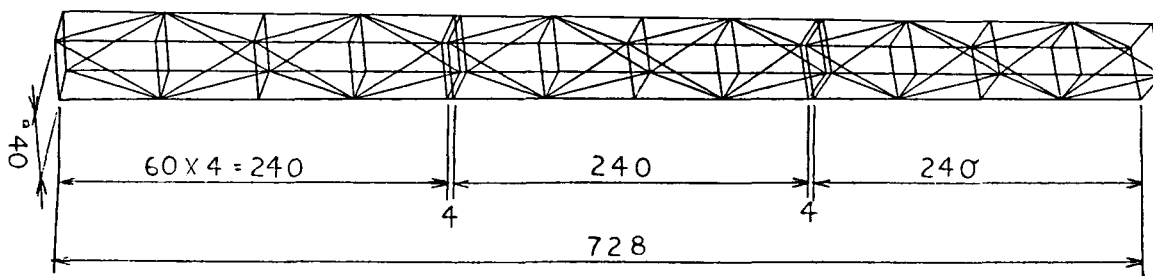


Figure 2. Schematic of a 12-bay space frame.

An appraisal of the computational mechanics methods is given in figure 3. These methods include the finite-element, boundary-element, and edge function methods (fig. 3).

FINITE ELEMENTS:

- TRIAL AND TEST FUNCTIONS ARE BOTH APPROXIMATE
- TRIAL AND TEST FUNCTIONS ARE, IN GENERAL, ALIKE - GALERKIN APPROACH
- IN SOME INSTANCES IT IS BEST TO HAVE TEST FUNCTIONS DIFFERENT FROM TRIAL FUNCTIONS - PETROV-GALERKIN APPROACH
- THE SOLUTION IS BOTH GLOBALLY AND LOCALLY APPROXIMATE
- VERSATILE OR ARBITRARY GEOMETRY, BOUNDARY CONDITIONS, SUITED FOR GLOBALLY APPROXIMATE NONLINEAR SOLUTIONS

BOUNDARY ELEMENTS:

- TEST FUNCTIONS ARE GLOBALLY EXACT FOR THE GIVEN LINEAR PROBLEM, OR AT LEAST FOR THE HIGHEST-ORDER DIFFERENTIAL OPERATOR OF THE PROBLEM
- TRIAL FUNCTIONS ARE APPROXIMATE (AT BOUNDARY ONLY FOR LINEAR PROBLEMS, AND IN INTERIOR ALSO FOR NONLINEAR PROBLEMS)
- THE SOLUTION IS BOTH LOCALLY AND GLOBALLY APPROXIMATE
- NOT AS VERSATILE AS THE FINITE-ELEMENT METHOD, BUT EXCELLENT FOR SOME SPECIFIC PROBLEMS

EDGE FUNCTION METHOD:

- TRIAL FUNCTIONS ARE GLOBALLY EXACT
- TEST FUNCTIONS ASSUMED ONLY AT BOUNDARY
- LIMITED TO LINEAR PROBLEMS POSED BY CLASSICAL DIFFERENTIAL EQUATIONS

Figure 3. Appraisal of computational mechanics methods.

In the most commonly used Galerkin finite-element approach in computational solid mechanics, the trial and test function spaces are identical and consist of simple piecewise continuous algebraic polynomials over each finite element, such that these functions and their appropriate-order derivatives (as dictated by the problem on hand) are continuous at the interelement boundaries. For problems of fourth or higher order, such as those of plates and shells, the development of finite elements has long been, and continues to be, plagued by the need for C^1 (or higher order) continuity at the interelement boundaries. However, the success of the finite-element method in structural mechanics is unparalleled and is mainly due to the intuitive and 'geometric' interpretation of the method. The method is versatile in its ability to deal with complicated structural assemblies, such as of beams, plates, and shells, of the type used in aerospace applications. The solutions obtained through the finite-element method may be classified, in general, as being both globally as well as locally approximate.

On the other hand, in linear and nonlinear solid mechanics, it is often possible to derive certain integral representations for displacements. A key ingredient which makes such derivations possible is the singular solution, in an infinite space, of the corresponding differential equation (in certain linear problems) or of the highest-order differential operator (in the nonlinear case, or even in the linear case when the full linear equation cannot be conveniently solved), for a 'unit' load applied at a generic point in the infinite space. When the problem is linear and the singular solution can be established for the complete linear differential equation of the problem, the aforementioned integral representations for displacements involve only boundary integrals of unknown trial functions and their appropriate derivatives. Such an integral representation, when discretized, leads to the so-called boundary-element method. Such pure boundary-element methods are possible in linear, isotropic, elastostatics, and in problems of static bending of linear elastic isotropic plates. On the other hand, as in such cases as (i) linear problems wherein the singular solutions cannot be established for the entire differential equations, (ii) anisotropic materials, and (iii) problems of large deformation and material inelasticity, the integral representations (if any) for displacements would involve not only boundary integrals but also interior-domain integrals of the trial functions and/or their derivatives. A discretization of such integral equations would lead not only to a simple boundary-element method but also to a sort of hybrid boundary/interior element method.

When asymptotic solutions to the governing differential equations of the problem are used as assumed trial functions, the interior residual error is zero, and only the boundary conditions need to be satisfied in a weighted residual method. Such an approach is called the edge-function method, but is limited mostly to linear problems. For further details, see references 1 through 4.

The surface-flow problems for current methods are noted in figure 4. Problems for the proposed method are also shown.

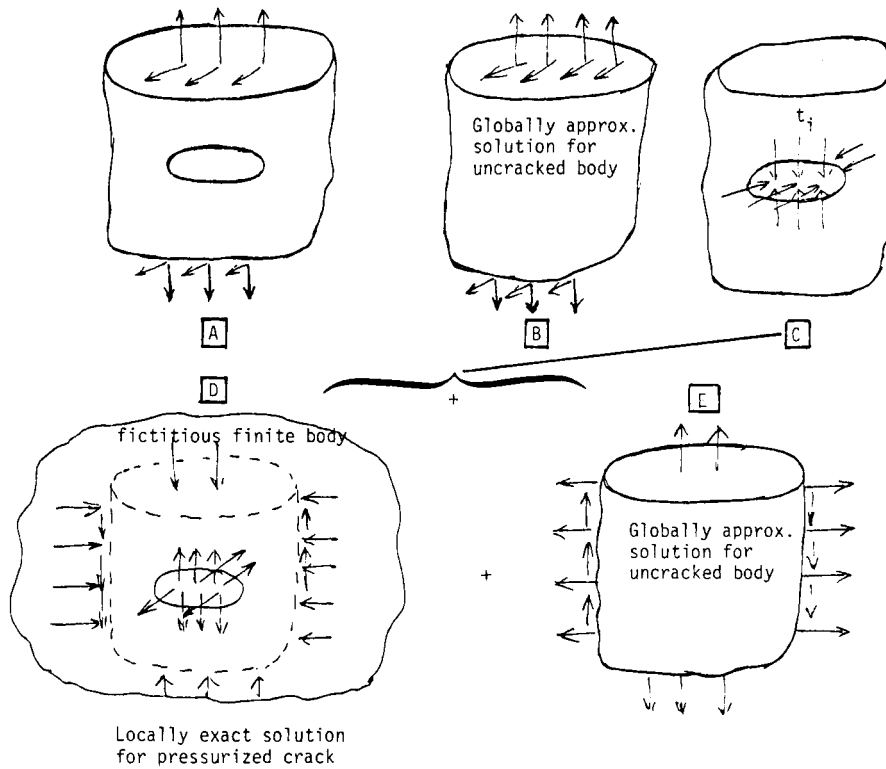
All Present Methods: Singular stress-state near the flaw border is modeled by locally approximate methods

- (I) Finite-Element Methods (singular elements)
 - Atluri & Kathiresan (refs. 5-10)
(Hybrid crack elements) 3-6,000 d.o.f.
 - Tracy, Barsoum, Newman & Raju (refs. 11-13)
(Distorted isoparametric elements and singular shape fn.) 5-10,000 d.o.f.
 - These are very expensive, but accommodate arbitrary geometries of structure and flaw.
- (II) Boundary-Element Methods (for locally approximate stress analysis and K-estimation from stress extrapolation)
 - Cruse (ref. 14), Heliot et al. (ref. 15).
 - Not suitable for 'thin' shells with flaws.
 - Still very expensive.
- (III) Line-Spring Method
 - Limited to simple geometries of structure and flaw.

Proposed Method:

- It is a GLOBALLY APPROXIMATE, but LOCALLY EXACT METHOD
- Singular stress-state near the flaw is NOT MODELED NUMERICALLY
- It is about 30 times cheaper than the singular finite-element method
- Details (Atluri & Nishioka (refs. 16-21) - several papers with varied examples)

Figure 4. Surface-flaw problems.



- Remarks:
1. Solution D: A rather complicated analytical solution (Atluri & Vijayakumar, Journal of Applied Mechanics, 1981) (ref. 22)
 2. Local solution due to crack-face traction alone (i.e., the Solution C) the source of singularity. The stresses due to this local solution decay very rapidly. Only one or two iterations are sufficient to obtain K-solutions with 1% accuracy.

About 30 times cheaper than the usual finite-element method for a typical problem such as flawed BWR nozzle.

Figure 5. Global (approximate) and local (exact) analyses of embedded flaws.

Some comments concerning the solution of surface flaws in finite bodies using the present procedure are in order (fig. 4). Since the analytical solution of an elliptical flaw embedded in an infinite solid is used as solution D, it is necessary to define the residual stresses over the entire crack-plane including the fictitious portion of the crack which lies outside of the finite body containing only a surface flaw (fig. 5). Moreover it is well known that the accuracy of the 'least-squares' type function-interpolation inside the interpolated region can be increased with the number of polynomial terms; however, the interpolating curve may change drastically outside of the region of interpolation. The optimum variation of pressure on the crack surface extended into the fictitious region should be as shown in figure 6. For in-depth discussions of a variety of surface problems and their solutions, see refs. 19-21, 23-28.

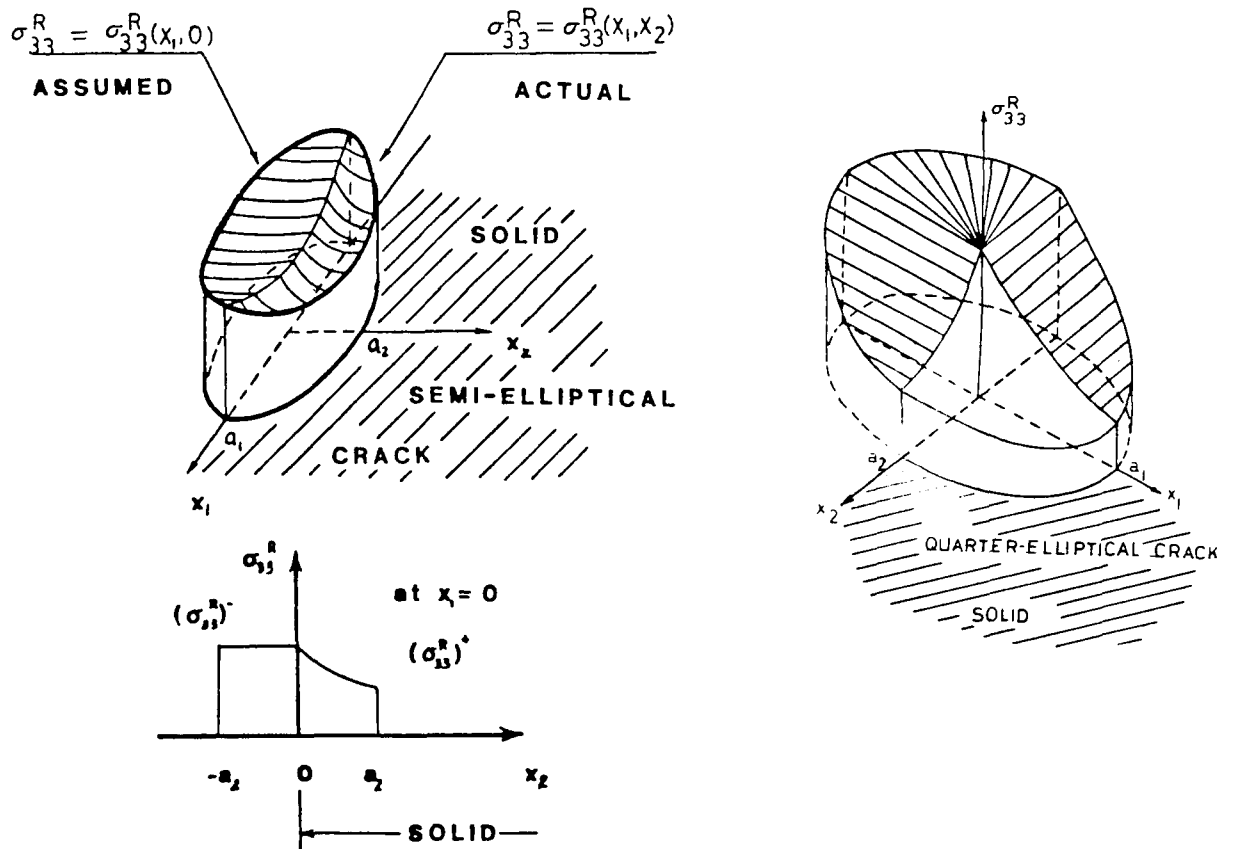


Figure 6. Postulated residual stress distributions on fictitious portions of an elliptical flaw, in the case of semi-elliptical or quarter-elliptical surface flaws.

For both 3-dimensional truss and frame members, explicit (locally exact) tangent stiffness matrices have been derived (fig. 7). Some effects of local (member) buckling on global (structural) behavior are illustrated in figures 8 and 9.

Truss Member:

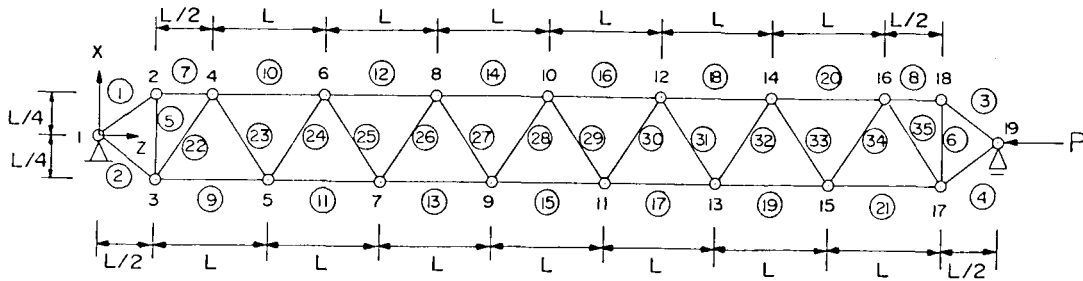
- Each member undergoes large displacement and large rigid rotation
- Member material is nonlinear
- Each member may buckle and become curved (what effect does it have on global stability?)

Frame Member:

- Concepts of 3-dimensional semi-tangential rotations employed
- Each member undergoes arbitrarily large rigid rotations and rigid displacements
- Bending-stretching coupling incorporated in each member
- Plastic-hinge method used to account for plasticity in each member
- Member forces: axial, shear, and bending-twisting moments

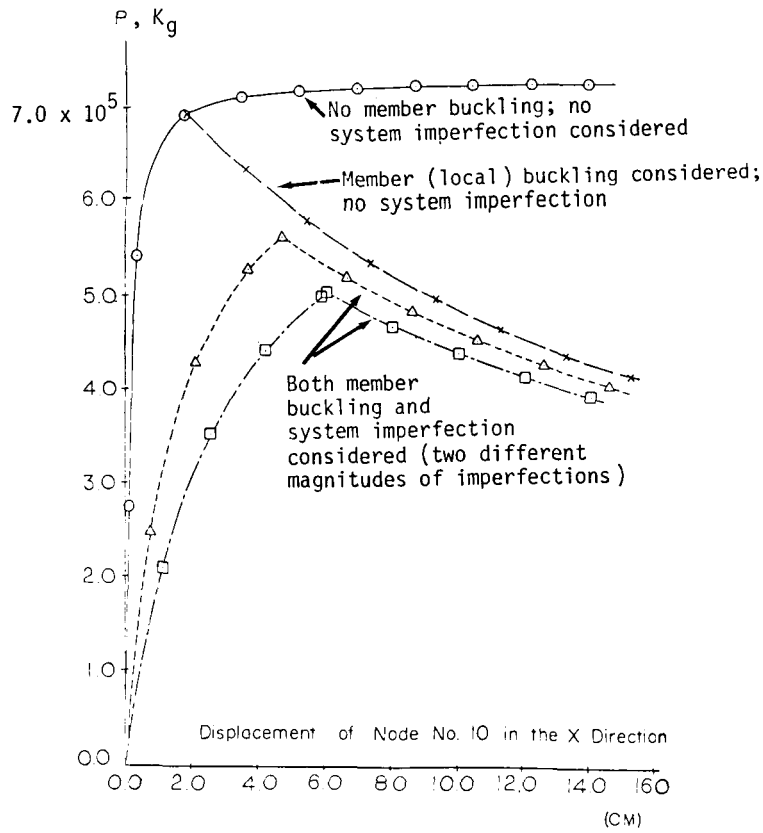
Kondoh and Atluri (refs. 29-30) and Kondoh et al. (refs. 31-32)

Figure 7. Space trusses and space frames.



$L = 66.04$ (cm)

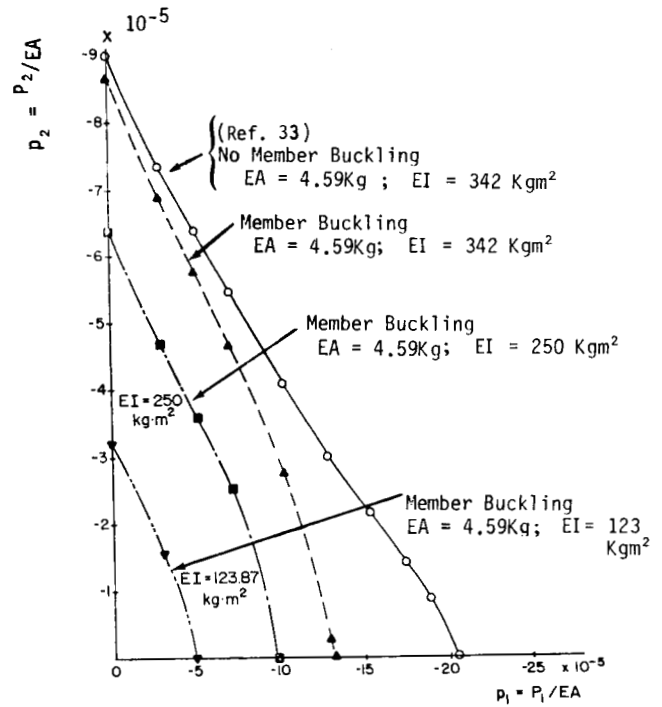
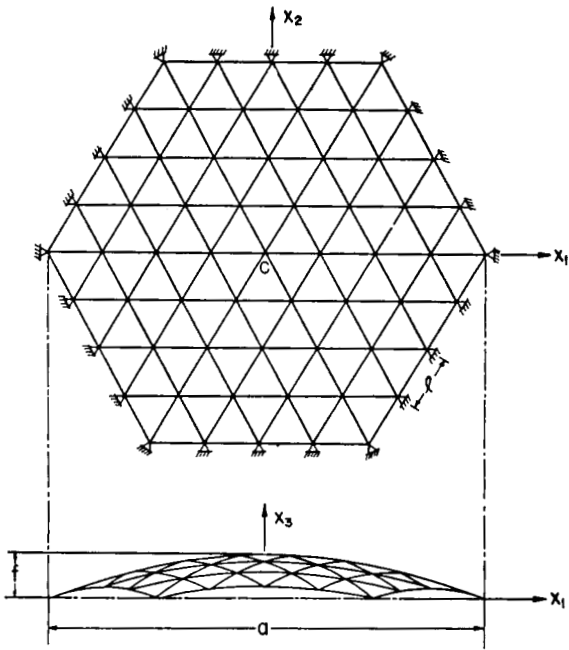
(a) Thompson's Strut



(b) Effect of local (member) buckling on global (structural) behavior.

Tangent stiffness of each member is exact in both the pre-buckled and post-buckled states of member (ref. 29)

Figure 8. Thompson's strut and effect of local buckling on global behavior.



- (a) Load System: (i) P_1 : Vertical point loads as all nodes;
(ii) P_2 : Vertical point loads at nodes in quadrants $x_1, x_2 > 0$

- (b) Stability boundary under loads P_1 and P_2

Figure 9. Study of the effect of member buckling on global (system) stability.

Examples of the efficiency of the global/local approach in analyzing frames are illustrated in figures 10 and 11. In figure 10, the classical problem of a two-bar frame is schematically illustrated. In the present approach, the tangent stiffness matrix of each member (represented by a single finite element) is derived from exact solutions of governing differential equations which account for the bending-stretching coupling. Thus, no "shape functions" are assumed in each element, and no numerical integrations are performed in forming the tangent stiffness matrix. The present numerical integrations are performed in forming the tangent stiffness matrix. The present numerical results are shown to agree excellently with those of Wood and Zienkiewicz (ref. 34), as well as the experimental results of Williams (ref. 35). However, Wood and Zienkiewicz use five finite elements to model each member of the frame.

In figure 11, the problem of plastic collapse of a frame is illustrated. Here again, the tangent stiffness matrix of each member (represented by a single finite element) is derived in closed form, accounting for large deformations, large rotations, and plasticity. A plastic-hinge method is used, and the progressive development of plastic hinges, at various load levels, is indicated in figure 11.

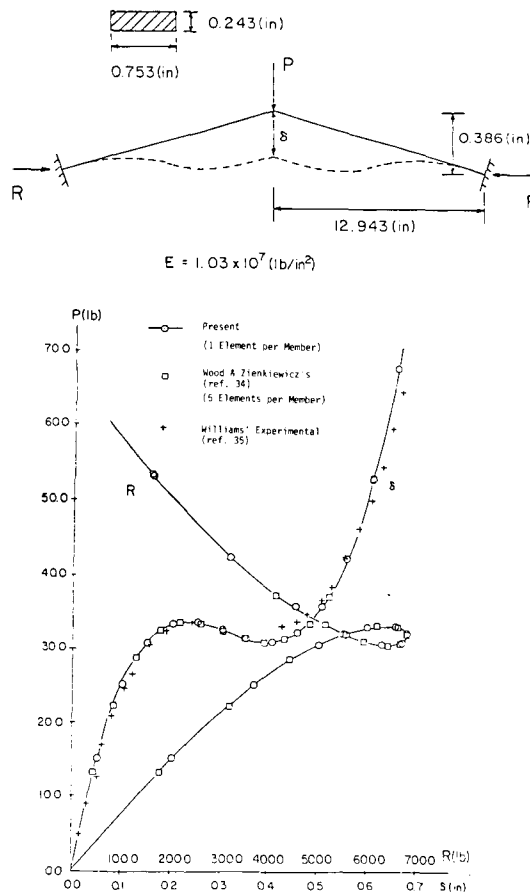
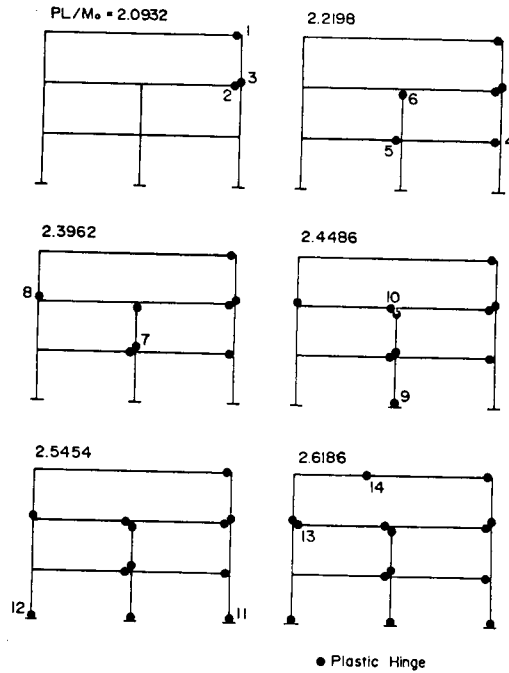
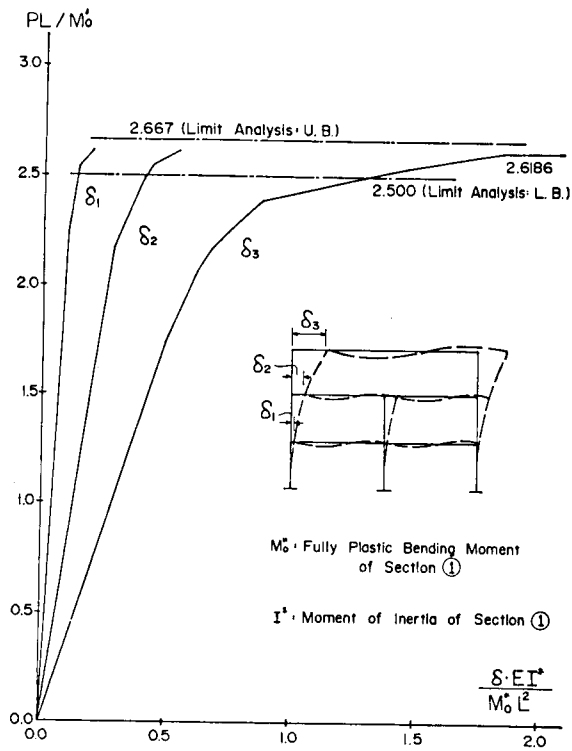


Figure 10. Variation of load-point displacement and support reaction with applied load in a two-bar frame. Tangent stiffness of each 3-D beam member undergoing large deformation, large rotation, and plasticity is exact. Locally exact solution (ref. 30).

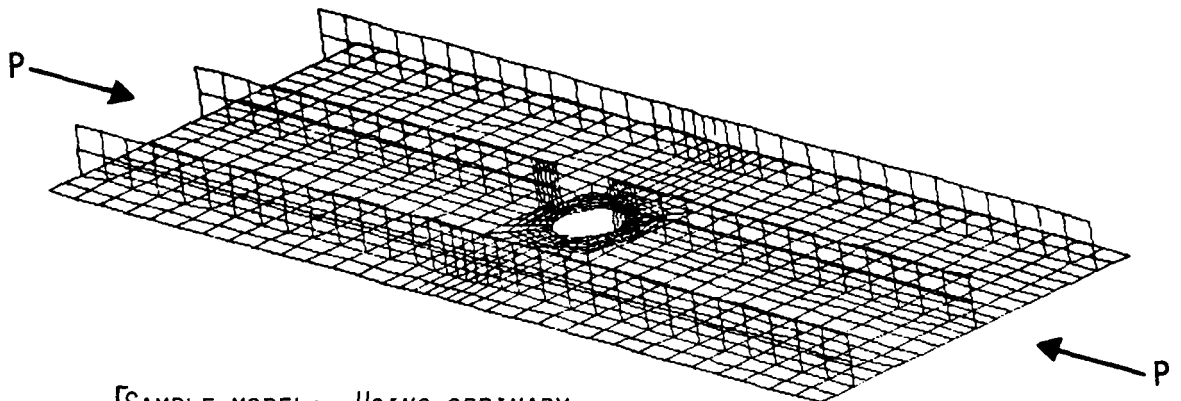


- TANGENT STIFFNESS OF EACH MEMBER UNDERGOING LARGE DEFORMATION, LARGE ROTATION AND PLASTICITY IS EXPLICIT AND EXACT. PLASTIC-HINGE METHOD USED.
- EXTENSION TO CRASH ANALYSIS OF FRAMES BEING STUDIED.

Figure 11. Plastic collapse of frame.

Figure 12 shows a problem of current interest in the analysis of stiffened composite plates. Issues involve the following: (1) stress concentrations near the hole in a composite laminate, (2) local buckling of stiffeners, (3) effect of geometric imperfections, (4) effect of discontinuities, and (5) three-dimensional effects and delaminations near the hole. An efficient globally approximate and locally exact approach could possibly include: (1) use of locally exact, laminated hole elements with embedded three-dimensional stress state (refs. 36 and 37), (2) use of locally exact stiffener elements as described earlier (ref. 32), (3) techniques for proper interacting of various elements, and (4) hole elements that can be improved by incorporating possible free-edge singularities in σ_{3i} .

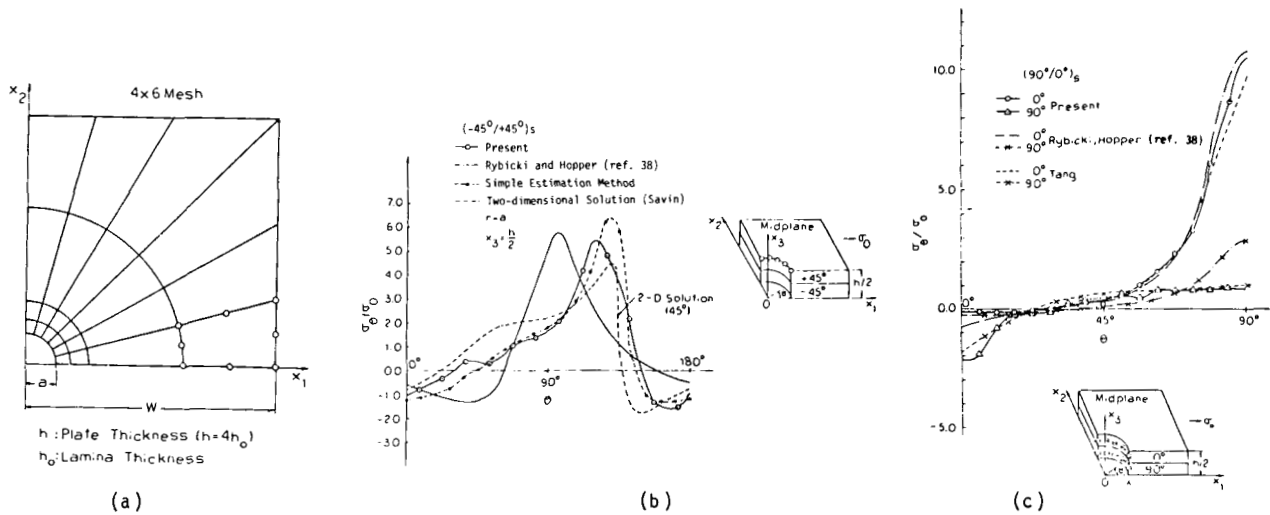
FOCUS PROBLEM



[SAMPLE MODEL: USING ORDINARY
(GLOBALLY & LOCALLY APPROXIMATE) FINITE ELEMENTS]

Figure 12. A stiffened laminated-composite panel with a hole.

Another example of the advantages of using a global/local approach is illustrated here in the problem of analysis of stresses near a hole in a laminated composite [two cases of $(-45/+45)_s$ and $(90/0)_s$ laminates are discussed]. Figure 13a shows a typical finite-element model with "special-hole elements" in which a 3-D asymptotic hole solution is embedded. Figures 13b and 13c illustrate the excellent accuracy obtained from the present approach, in comparison with a fully 3-D finite-element solution of Rybicki and Hopper (ref. 38). The present solution is, however, an order of magnitude less expensive. Further details are given in references 36 and 37.



TYPICAL FEM MODEL OF A LAMINATE WITH HOLE, 3-D ASYMPTOTIC "HOLE-SOLUTIONS" EMBEDDED IN ELEMENTS NEAR THE HOLE (Ref. 36)

STRESS AROUND A HOLE IN $(-45/+45)_s$ LAMINATE

STRESS AROUND A HOLE IN $(90/0)_s$ LAMINATE

Figure 13. Analysis of stress state near a hole in laminated composites.

The following conclusions and recommendations are given.

- Hybrid analytical/numerical methodologies should be explored
- Simplified analysis procedures for elasto-plastic should be considered (Dynamic response calculations should be studied (some benchmark problems essential))
- Constitutive models badly need improvement
- Methods of coupling of problem-specific methodologies for use in general purpose programs should be explored
- Trends to treat structural mechanics problems as continuum mechanics problems should be critically reviewed; the knowledge base in structural mechanics should be fruitfully utilized
- Attempts to bridge the gap between micromechanics and macromechanics of heterogeneous (composite) media through computational mechanics should be explored
- Computational stochastic structural analysis methods should be developed
- Algorithms for new computing systems (MIMD) should be explored
- Expert systems, (?)

NASA's role should be to provide:

- Increased research support to universities
- Predoctoral NASA fellowships (up to 20K per year, tax-free) that could be awarded to attract the best students
- Long-range funding to properly plan and sustain high-quality research efforts
- Increased access to supercomputers
- Frequent visits to NASA facilities by graduate students to participate in laboratory testing. University facilities in this area are scarce; students in computational mechanics should get some first-hand experience in experimental mechanics

REFERENCES

1. Atluri, S.N. and Kathiresan, K., "Influence of Flaw Shapes on Stress Intensity Factors for Pressure Vessel Surface Flaws and Nozzle Corner Cracks", Journal of Pressure Vessel Technology, Vol. 102, August 1980, pp. 78-286.
2. Atluri, S.N. (Editor and Contributor to Part II, Finite Element Methods), Finite Element Handbook, McGraw-Hill Book Co., 1200 pp. (In Press).
3. Atluri, S.N., "Computational Solid Mechanics (Finite Elements and Boundary Elements): Present Status and Future Directions", (Keynote Paper) in Proc. 4th International Conference on Applied Numerical Modeling (Eds: H.M. Hsia, et al.), pp. 19-37, 1984.
4. O'Donoghue, P.E. and Atluri, S.N., "Control of Dynamic Response of a Continuum Model of a Large Space Structure", AIAA 26th Structures, Structural Dynamics & Materials Conference, Orlando, Florida, April 15-17, 1985.
5. Atluri, S.N., Kathiresan, K., and Kobayashi, A.S., "Three-Dimensional Linear Fracture Mechanics Analysis by a Displacement-Hybrid Finite-Element Model", Trans. of the 3rd International Conference on Structural Mechanics in Reactor Technology, London, Vol. 5, Part L, Paper No. L-7/3, September 1975, 13 pp.
6. Atluri, S.N. and Kathiresan, K., "On a 3-D 'Singularity Element' for Computation of Combined Mode Stress Intensities", Advances in Engineering Science, Vol. 1, November 1976, pp. 267-274.
7. Atluri, S.N., Kathiresan, K., Kobayashi, A.S., and Nakagaki, M., "Inner Surface Cracks in an Internally Pressurized Cylinder Analyzed by a Three-Dimensional Displacement-Hybrid Finite Element Method", Pressure Vessel Technology, Part II - Materials and Fabrication, ASME, 1977, pp. 527-533.
8. Atluri, S.N. and Kathiresan, K., "Outer and Inner Surface Flaws in Thick-Walled Pressure Vessels", Trans. of 4th International Conference on Structural Mechanics in Reactor Technology, Paper G 5/4, 1977, 13 pp.
9. Atluri, S.N. and Kathiresan, K., "Stress Analysis of Typical Flaws in Aerospace Structural Components Using 3-D Hybrid Displacement Finite Element Method", Proceedings of 19th AIAA/ASME/SAE Structures, Structural Dynamics and Materials Conference, Bethesda, Maryland, 1978, pp. 340-351.
10. Atluri, S.N. and Kathiresan, K., "3-D Analysis of Surface Flaws in Thick-Walled Reactor Pressure Vessels Using Displacement-Hybrid Finite Element Method", Nuclear Engineering & Design, Vol. 51, No. 2, January 1979, pp. 163-176.
11. Tracy, D.M., "Finite Elements for Three-Dimensional Elastic Crack Analysis", Nuclear Engineering & Design, Vol. 16, 1977, pp. 282-290.
12. Barsoum, R.S., "On the Use of Isoparametric Finite Elements in Linear Fracture Mechanics", International Journal for Numerical Methods in Engineering, Vol. 10, No. 1, 1976, pp. 25-37.

13. Newman, J.C., Jr. and Raju, I.S., "Stress-Intensity Factor Equations for Cracks in Three-Dimensional Finite Bodies Subjected to Tension and Bending Loads", (S.N. Atluri, Ed.) Computational Method in the Mechanics of Fracture, North-Holland Publishing Co. (In Press).
14. Cruse, T.A., "Boundary-Integral Equation Method for Three-Dimensional Elastic Fracture Mechanics", AFOSR-TR-75-0813, U.S. Air Force, 1975, 52 pp.
15. Heliot, J., Labbens, R.C., and Pellissier-Tannon, A., "Semi-Elliptical Cracks in a Cylinder Subjected to Stress Gradients", in Fracture Mechanics, ASTM STP 677, 1979, pp. 341-364.
16. Atluri, S.N. and Nishioka, T., "Analytical Solution for Embedded Elliptical Cracks, and Finite Alternating Method for Elliptical Surface Cracks, Subjected to Arbitrary Loadings", Engineering Fracture Mechanics, Vol. 17, No. 3, 1982, pp. 247-268.
17. Atluri, S.N. and Nishioka, T., "Integrity Analyses of Surface-Flawed Aircraft Attachment Lugs: A New, Inexpensive, 3-D Alternating Method", Proceedings of 22nd AIAA/ASME/ASCE/AHS Structures and Material Conference, New Orleans, Louisiana, 1982, pp. 287-301.
18. Atluri, S.N. and Nishioka, T., "Analysis of Surface Flaw in Pressure Vessels by a New 3-Dimensional Alternating Method", Aspects of Fracture Mechanics in Pressure Vessels and Piping, PVP, Vol. 58, ASME, New York, 1982, pp. 17-37. Also Journal of Pressure Vessel Technology, ASME Vol. 104, 1982, pp. 299-307.
19. Atluri, S.N. and Nishioka, T., "Stress Intensity Factor Variation with Loading and Crack Depth for Long Flaws of Semi-Elliptical Shape", Advances in Aerospace Structures, AD Vol. 2, ASME, 1982, pp. 47-50.
20. Atluri, S.N. and Nishioka, T., "An Inexpensive 3-D Finite Element-Alternating Method for the Analysis of Surface Flawed Aircraft Structural Components", AIAA Journal, Vol. 21, No. 5, 1983, pp. 749-758.
21. Atluri, S.N. and Nishioka, T., "Multiple Semi-Elliptical Surface Cracks in Pressure Vessels", Journal of Pressure Vessel Technology, Trans. ASME, Vol. 54, 1982, pp. 612-619.
22. Atluri, S.N. and Vijayakumar, K., "An Embedded Elliptical Crack, in an Infinite Solid, Subject to Arbitrary Crack-Face Traction", Journal of Applied Mechanics, Vol. 103, No. 1, 1981, pp. 88-96.
23. O'Donoghue, P.E., Nishioka, T., and Atluri, S.N., "Multiple, Coplanar, Embedded Elliptical Cracks, in an Infinite Solid Subject to Arbitrary Crack Face Traction", International Journal of Numerical Methods in Engineering, Vol. 21, 1985, pp. 437-449.
24. Atluri, S.N. and Nishioka, T., "Computational Methods for Three-Dimensional Problems of Fracture", in Computational Methods in the Mechanics of Fracture (S.N. Atluri, Editor), North-Holland (In Press).
25. O'Donoghue, P.E., Nishioka, T., and Atluri, S.N., "Multiple Surface Cracks in Pressure Vessels", Engineering Fracture Mechanics, Vol. 20, No. 3, 1984, pp. 545-560.

26. Atluri, S.N., "Computational and Theoretical Studies on Dynamic Fracture Mechanics and Three-Dimensional Crack Problems", Keynote Lecture, Proceedings of ICF International Symposium on Fracture Mechanics, Science Press, Beijing, 1983, pp. 1-17.
27. O'Donoghue, P.E., Nishioka, T., and Atluri, S.N., "Analysis of Interaction Behavior of Surface Flaws in Pressure Vessels", Journal of Pressure Vessels Technology, Trans. ASME, 1984.
28. Punch, E.F. and Atluri, S.N., Development and Testing of Stable, Invariant, Isoparametric Curvilinear 2- and 3-D Hybrid Stress Elements", Computer Methods in Applied Mechanics and Engineering, Vol. 47, 1984, pp. 331-356.
29. Kondoh, K. and Atluri, S.N., "Influence of Local Buckling on Global Instability: Simplified, Large Deformation, Post-Buckling Analyses of Plane Trusses", Computers and Structures, Vol. 21, No. 4, 1985, pp. 613-628.
30. Kondoh, K. and Atluri, S.N., "A Simplified Finite Element Method for Large Deformation, Post-Buckling Analyses of Large Frame Structures, Using Explicitly Derived Tangent Stiffness Matrices", International Journal of Numerical Methods in Engineering, Vol. 23, No. 1, 1986, pp. 69-90.
31. Kondoh, K., Tanaka, K., and Atluri, S.N., "LSS-NLAP: A Program for Simplified Nonlinear Analysis of Large Space-Trusses and Frames, Using Explicitly Derived Tangent Stiffnesses, and Accounting for Local Buckling", Report to Air Force Wright Aeronautical Laboratories, 1985, 171 pp, USAF, WPAFB.
32. Tanaka, K., Kondoh, K., and Atluri, S.N., "Instability Analysis of Space Trusses Using Exact Tangent-Stiffness Matrices" in Finite Element Analysis & Design, North-Holland, Vol. 1, No. 4, 1985, pp. 291-312.
33. Noor, A.K. and Peters, J.M., "Instability Analysis of Space Trusses", Computer Methods in Applied Mechanical Engineering, Vol. 40, 1983, pp. 199-218.
34. Wood, R.D. and Zienkiewicz, O.C., "Geometrically Nonlinear Finite Element Analysis of Beams, Frames, Arches, and Axisymmetric Shells", Computers and Structures, Vol. 7, 1977, pp. 725-735.
35. Williams, F.W., "An Approach to the Nonlinear Behavior of Members of a Rigid Jointed Framework With Finite Deflections", Quant. J. Mech. Appl. Math., Vol. 17, No. 4, 1964, pp. 451-469.
36. Atluri, S.N. and Nishioka, T., "Stress Analysis of Holes in Angle-Ply Laminates: An Efficient Assumed Stress 'Special-Hole-Element', Approach and a Simple Estimation Method", Computers and Structures, Vol. 15, November 1982, pp. 135-147.
37. Atluri, S.N. and Nishioka, T., "A Simple Estimation Method of Stress Intensity Factors for Through Cracks in Angle-Ply Laminates", Engineering Fracture Mechanics, Vol. 16, No. 4, 1982, pp. 573-583.
38. Rybicki, E.F. and Hopper, A.T., "Analytical Investigation of Stress Concentrations Due to Holes in Fiber Reinforced Plastic Laminated Plates: Three-Dimensional Models", AFML-TR-73-100, Battelle Columbus Labs, 1973.

ON COMPUTATIONAL SCHEMES FOR GLOBAL-LOCAL
STRESS ANALYSIS

J. N. Reddy
Virginia Polytechnic Institute and State University
Blacksburg, Virginia

1. INTRODUCTION

This paper primarily deals with an overview of global-local stress analysis methods and associated difficulties and recommendations for future research. The phrase global-local analysis is understood to be an analysis in which some parts of the domain or structure are identified, for reasons of accurate determination of stresses and displacements or for more refined analysis than in the remaining parts. The parts of refined analysis are termed local and the remaining parts are called global. Typically local regions are small in size compared to global regions, while the computational effort can be larger in local regions than in global regions.

2. CONTENTS

This paper is divided into the following parts:

- Motivation-Problems (problems that motivated global-local analysis)
- Common Features
- Focus Problem
- Analysis Methods
- Global-Local Approaches
- Example Problem
- Conclusions and Recommendations

3. MOTIVATION PROBLEMS

The following stress analysis problems, among many others, motivate us to use global-local approaches:

- Free-Edge Stress Concentration in Laminates
- Contact Stress Problems
- Impact
- Fracture Mechanics
- Unbounded-Domain Problems

Almost all laminated composite structural elements have free edges at the boundaries (including holes) of the elements. It is well known that the transverse normal and shear stresses are very large at the edges (more precisely, the stresses are large within a distance of the order of thickness of the laminate from the free edge). While the classical laminate theory is adequate to describe the behavior of the laminate everywhere except in the "boundary layer" in which the transverse normal and shear stresses are large, a refined theory, often the 3-D elasticity theory, is needed to describe the state of stress near the edges.

Contact stress problems (for example, bolted and bonded joints, tire contact, and metal-forming problems) require the use of a special theory that accounts for appropriate constitutive laws and friction and allows for slip, slide, and separation of the mating parts in the contact regions. Elsewhere, appropriate elasticity theory can be used.

Impact of two solid bodies can be modelled by the use of one theory in the vicinity of contact and by another theory elsewhere. Since the stresses are much larger in the contact region than elsewhere, more refined theory and analysis are required in contact regions. Of course, the theory and analysis used depend on the type of structure, loading and deformation.

Structures containing cracks, whether formed during manufacturing or service, require special treatment of stress fields around cracks, often using 3-D stress analyses and/or nonlinear fracture mechanics theories, while the linear elastic fracture mechanics theory is adequate away from the cracks.

Problems involving unbounded regions (for example, soil mechanics and earthquake engineering) are by their nature divided into local and global regions. Global regions, in theory, can be infinite but in practice they are finitely large, and less refined theory and/or analysis is used to determine the stress field and other pertinent information.

4. COMMON FEATURES

The motivating problems listed previously share certain common physical features that are significant from the modelling and analysis points of view. The following list provides some of these features:

- Stress Concentration (large local gradients)
- Three-Dimensional State of Stress
- Large Rotations/Strains
- Local Discontinuities (holes, discontinuous fibers, etc.)
- Material Nonlinearities (nonlinear elasticity, plasticity, etc.)

A global-local stress analysis should account for all features that are present in the problem. Of course, some of those features are not to be included in the global model.

5. FOCUS PROBLEM

The focus problem identified by NASA Langley Research Center is a blade-stiffened panel with a discontinuous stiffener. The problem has the following physical features:

- Geometric Discontinuities
- Local Stress Gradients
- Eccentric Loading
- Large Displacements
- Free Edges

We shall return to this problem later to discuss the global-local analysis approach.

6. ANALYSIS METHODS

6.1 COMMON APPROACHES

The commonly used analysis methods for structural problems are

- Classical (or Analytical) Methods
- Classical Variational Methods
- Finite-Difference Method
- Finite-Element Method
- Boundary Element Method

Some noted advantages and disadvantages of these methods are outlined next.

6.2 CLASSICAL AND VARIATIONAL METHODS

The classical method of solving problems exactly is the best there is. However, most practical problems (which have irregular geometries, anisotropic materials, discontinuities, geometric and/or material nonlinearities, etc.) do not admit exact solutions by the classical approach.

The classical variational (e.g., Ritz, Galerkin and weighted-residual) methods yield continuous solutions throughout the domain, giving high resolution of displacements and stresses. They are computationally efficient for a given problem. For a given order of approximation, previously computed (for lower order approximation) matrix coefficients can be used. These methods, however, have two major shortcomings: (i) the approximation functions are not easy and are often impossible to

construct for most practical structural problems; and (ii) the variational methods cannot be implemented on a computer for the analysis of a class of problems because the resulting algebraic equations depend on the approximation functions, which in turn depend on a specific problem.

6.3 FINITE-DIFFERENCE METHOD (FDM)

The finite difference method is simple in formulation (based on the representation of derivatives of a function in terms of a finite Taylor's series expansion) and easy to implement on the computer. The method dominated the field of numerical methods until the sixties, when the finite-element method gained popularity, especially in solid and structural mechanics. The disadvantages of the finite-difference method include: difficulty in representing complex geometries, inexact representation of boundary conditions on non-straight boundaries, and difficulty in developing higher-order approximations. Because of these difficulties the method does not lend itself for general-purpose code development. The method is seldom used for spatial approximations in structural mechanics problems.

6.4 FINITE-ELEMENT METHOD

The finite-element method overcomes the shortcomings of the classical variational methods. This approach is systematic (modular) and natural and allows an accurate representation of complex geometries. Higher-order approximations are easy to use without changing the modular structure of the approach. The method is ideally suited for general purpose and computer program development. The disadvantages, compared to other computing methods, are the large formulative and computational efforts. The finite-element method is the most frequently used numerical method in structural analysis. It is now a key component of any mechanical CAD/CAM system.

6.5 BOUNDARY ELEMENT METHOD (BEM)

The finite-difference and finite-element methods can be classified as domain methods because they involve approximations of the entire domain. The boundary element method, also known as the boundary integral method (BIE), seeks approximations only on the boundary of the domain by converting the governing differential equations to integrals over the boundary of the domain. The dimensionality of the problem is thereby effectively reduced by one. Because the interior of the domain is not approximated, the computational time involved is less ($BEM/FEM \approx \frac{1}{n}$, where $n \times n$ is the finite-element mesh). The BEM offers continuous interior modelling within the solution domain, giving high resolution of displacements and stresses. The method is unsuitable for problems requiring information at a large number of internal points. Application of BEM to nonlinear problems and problems with discontinuities is not fully established.

6.6 CONCLUSIONS

In conclusion, the general application of the finite-element method to structural problems is unmatched to date. Structural problems, because of the modular nature of FEM, are better modelled by FEM. A suitable combination of FEM and BEM can be advantageous in some problems (e.g., unbounded-domain problems).

7. GLOBAL-LOCAL APPROACHES

7.1 MODELS AND METHODS

In formulating a given problem, either the same theory is used throughout or a refined theory is used locally and a less refined theory globally.

The global-local analysis methods can be FEM throughout, FEM and classical solution, or BEM and FEM. When the same theory and FEM are used throughout, it is understood that locally special elements are used (e.g., friction element, interface element with sliding, or elements which allow opening and closing along element interfaces). Classical solutions are available, for example, for infinite plates with holes. The solution is not valid far away from the hole if the plate is long but not infinite. In such cases, the FEM can be used globally and the classical solution can be used locally. For soil mechanics and earthquake engineering problems, a combination of BEM and FEM proves computationally efficient. In some situations experimental methods globally and computational methods locally are recommended.

7.2 SOME EXAMPLES

Some example problems that require global-local analysis are listed here.

- Free-edge stress analysis of laminates
- Contact stress problems
- Stress analysis of structures with discontinuities
- A blade-stiffened panel with a discontinuous stiffener - the focus problem

As mentioned earlier, free-edge stress problem requires a refined theory near free edges. For example, the classical laminate theory globally and either quasi-3D or full 3-D theory locally (depending on the lamination scheme, geometry and loading) can be used to analyze the problem. The problem will be discussed in more detail later.

In bolted joint problems, an experimental technique such as the Moiré interferometry can be used to determine the surface displacements (and hence strains and stresses) and the finite-element method can be used to determine the interior displacement and stress fields.

In the stress analysis of plates with holes, one can use a laminate theory away from holes and 3-D elasticity theory around the hole, and use the finite-element method to model the entire problem.

In the case of the focus problem, which has all features that are present in the examples discussed above, the global theory should be a shear deformation plate theory (2D) with the Von Karman geometric non-linearity and the local theory can be the fully 3D laminate theory. The finite-element method should be used throughout. When FEM is used locally, the fully 3D elements or 3D degenerate elements can be used.

7.3 DIFFICULTIES

In using global-local approaches, we face some difficulties. Some of these are

- Interfacing between regions
- Interfacing between methods
- Selection of regions
- Changing regions and interfaces

When the finite-element method is used, the elements used globally and locally can be different. Then it is important to have compatibility of the nodal degrees of freedom at the interface of the elements. A special interface element might be needed in some situations. When two different methods are used, the unknowns in the two methods should be the same. Selection of the local and global regions depends on the physical features and accuracy desired. In some cases, the regions might have to be determined only after a preliminary analysis. The global and local regions can change during the history of deformation/ loading. For example, in elastic-plastic analysis, the plastic zones are unknown a priori and they change with loading.

8. EXAMPLE PROBLEM

Here we briefly discuss the free-edge stress problem in symmetric laminates. Figure 1 shows the laminate geometry, loading, and the domain modelled. Because of the assumed symmetry of the lamination about the midplane and the constant straining along the x-axis, the displacement field can be approximated (ref. 1) as

$$u = U_0 x + U(y, z)$$

$$v = V(y, z)$$

$$w = W(y, z)$$

where U_0 is a constant, K.

The displacement field is three dimensional but it leads, when substituted into the Navier equations of equilibrium, to three partial differential equations in two independent variables, y and z .

It is well known that the transverse normal stress, σ_z , is very large (unbounded) near the free edge. To reduce its magnitude a cap is used on the free edge. The effect of the cap on the stress distribution σ_z is investigated. The finite-element method with the four-node bilinear rectangular element is used to model the computational domain. A refined mesh is used near the free edge and in the cap.

Figures 2 and 3 show the distribution of the transverse normal stress σ_z along the width of the laminate for $[0^\circ/90^\circ]_5$ and $[45^\circ/-45^\circ]_5$ laminates, respectively, ($E_1 = 137.89$ GPa, $E_2 = E_3 = 14.48$ GPa, $G_{12} = G_{13} = G_{23} = 5.86$ GPa, $\nu_{12} = \nu_{13} = \nu_{23} = 0.21$). Results for both capped and uncapped laminates are presented (for $k = 0.001$, $b = 25.4$ cm, $h = 2.54$ cm and thickness of the cap, $t = 0.08$ cm). We observe that the stress is essentially zero inside the laminate but has quite a large magnitude within a distance of $y/b = 0.1$ (one-tenth of the width) from the free edge. Hence a laminate theory is sufficient to model the interior, while the quasi-3D can be used to model the free-edge stress field. The effect of the free-edge reinforcement (i.e., cap) on the stress magnitude is significant; the magnitude is reduced to less than one-third of that without cap.

For a more detailed and complete stress distribution near the free edge of a more general laminate (e.g., without symmetry about the midplane), a three-dimensional model is needed.

9. CONCLUSIONS AND RECOMMENDATIONS

9.1 AREAS NEEDING SUPPORT

A review of the literature shows there are very few cases of global-local analyses of structural problems involving the "physical features" discussed earlier. It is recommended that the following areas of global-local approaches be investigated:

- Global-local analysis of problems with "common features" outlined earlier
- Investigation and development of interface elements
- Feasibility of BEM as a computational tool for nonlinear problems and its interface with FEM
- Development of adaptive mesh refinements and time-stepping algorithms
- Exploitation of the vector and parallel processor computers for efficient structural analysis

- Finite-element calculations
- Solution of equations
- Eigenvalue computations

The use of parallel processors can dictate the solution procedures, for example, iterative methods over one-step methods.

9.2 NASA'S INVOLVEMENT

NASA (CSM) should be involved in the global-local analysis development because of the tremendous impact this field has on computational mechanics applied to space structures. In particular, NASA should undertake the following tasks in the global-local analysis area:

- Support individual grants (as opposed to large group grants)
- Collaborate with university faculty and graduate students by identifying specific problem areas and providing computational time and scientific advice
- Give graduate student residencies, during which students spend a few weeks (perhaps the summers) at NASA
- Conduct workshops (say, once in two years) to bring the latest developments for critical evaluation and to set future directions.

ACKNOWLEDGMENTS

This paper was prepared via the support of the Structural Mechanics Branch of NASA Langley Research Center (NAG-1-459).

REFERENCE

Heyliger, P. R. and Reddy, J. N., "Reduction of Free-Edge Stress Concentration in Symmetric Laminates," Virginia Polytechnic Institute and State University, Blacksburg, VA, CCMS Report 85-01, January 1985.

BIBLIOGRAPHY

Arnold, D. N., Babuska, I. and Osborn, J., "Finite Element Methods: Principles for Their Selection," Computer Methods in Appl. Mech. & Engng., Vol. 45, pp. 57-96, 1984.

Atluri, S. N. GALLAGHER, R. H., and Zienkiewicz, O. C., Hybrid and Mixed Finite Element Methods, Wiley, Chichester (U.K.), 1983.

- Banerjee, P. K. and Butterfield, R., Boundary Element Methods in Engineering Science, McGraw-Hill, London, 1981.
- Brebbia, C. A., The Boundary Element Method for Engineers, Pentech Press, London, 1980.
- Buzbee, B. L. and McKnight, A. L., "Impact of Vector Processors on Algorithms," Los Alamos Scientific Laboratory, Los Alamos, NM 87545, LA-8123-MS, 1979.
- Calahan, D. A., "Exploiting Parallelism in Designing Algorithms," Basic Quest. Des. Theory, (Symposium, New York, 1974), pp. 285-302, 1974.
- Chen, S. C., Kuck, D. J. and Sameh, A. H., "Practical Parallel Band Triangular System Solvers," ACM Trans. Math. Software, Vol. 4, pp. 270-77, 1978.
- Conrad, V. and Wallach, Y., "Iterative Solution of Linear Equations on a Parallel Processor System," IEEE Trans. Computers, Vol. 26, pp. 838-47, 1977.
- Csanky, L., "Fast Parallel Matrix Inversion Algorithms," SIAM J. Comput., Vol. 5, pp. 618-23, 1976.
- Durnier, A., "A Theory of Friction," Int. J. Solids & Struct., Vol. 20, No. 7, pp. 637-647, 1984.
- Fenner, R. T., "The Boundary Integral Equation (Boundary Element) Method in Engineering Stress Analysis," J. Strain Analysis, Vol. 18, No. 4, 1983.
- Feng, T., (ed.), Parallel Processing, Springer-Verlag, New York, 1975.
- Gadala, M. S. and Oyavas, G. AE., "Numerical Solutions of Nonlinear Problems of Continua-I (Survey of Formulation Methods and Solution Techniques)," Computers and Structures, Vol. 19, No. 5/6, pp. 865-877, 1984.
- Heller, D., "A Survey of Parallel Algorithms in Numerical Linear Algebra," SIAM Rev., Vol. 20, pp. 740-77, 1978.
- Knight, Jr., N. F. and Starnes, Jr., J. H., "Postbuckling Behavior of Selected Curved Stiffened Graphite-Epoxy Panels Loaded in Axial Compression," 26th AIAA/ASME/ASCE/AHS Structures, Structural Dynamics and Materials Conference, AIAA Paper No. 85-0768-CP, Orlando, FL., 1985.
- Matthies, H. and Strang, G., "The Solution of Nonlinear Finite Element Equations," Int. J. Numer. Meth. in Engrng., Vol. 14, pp. 1613-1626, 1979.
- Ortega, J. M. and Voigt, R. G., "Solution of Partial Differential Equations on Vector and Parallel Computers," NASA CR-172500, Jan. 1985.
- Pagano, N. J. and Soni, S. R., "Global-Local Laminate Variational Model," Int. J. Solids and Struct., Vol. 19, No. 1, pp. 207-228, 1981.
- Pipes, R. B. and Pagano, N. J., "Interlaminar Stresses Composite Laminates Under Uniform Axial Extension," J. Comp. Materials, Vol. 4, pp. 538-548, 1970.

- Reddy, J. N., An Introduction to the Finite Element Method, McGraw-Hill, New York, 1984.
- Reddy, J. N., Energy and Variational Methods in Applied Mechanics, Wiley, New York, 1984.
- Rybicki, E. F., "Approximate Three-Dimensional Solutions for Symmetric Laminates Under Inplane Loading," J. Comp. Materials, Vol. 5, pp. 354-360, 1971.
- Traub, J. F. (ed.), Complexity of Sequential and Parallel Numerical Algorithms, Academic Press, New York, 1973.
- Whitcomb, J. D., Raju, L. S. and Goree, J. G., "Reliability of the Finite Element Method for Calculating Free-Edge Stresses in Composite Laminates," Computers and Structures, Vol. 15, No. 1, pp. 23-37, 1982.
- Zienkiewicz, O. C., Gago, J. P., De, S. R., and Kelly, D. W., "The Hierarchical Concept in Finite Element Analysis," Computers and Structures, Vol. 16, No. 1-4, pp. 53-65, 1983.

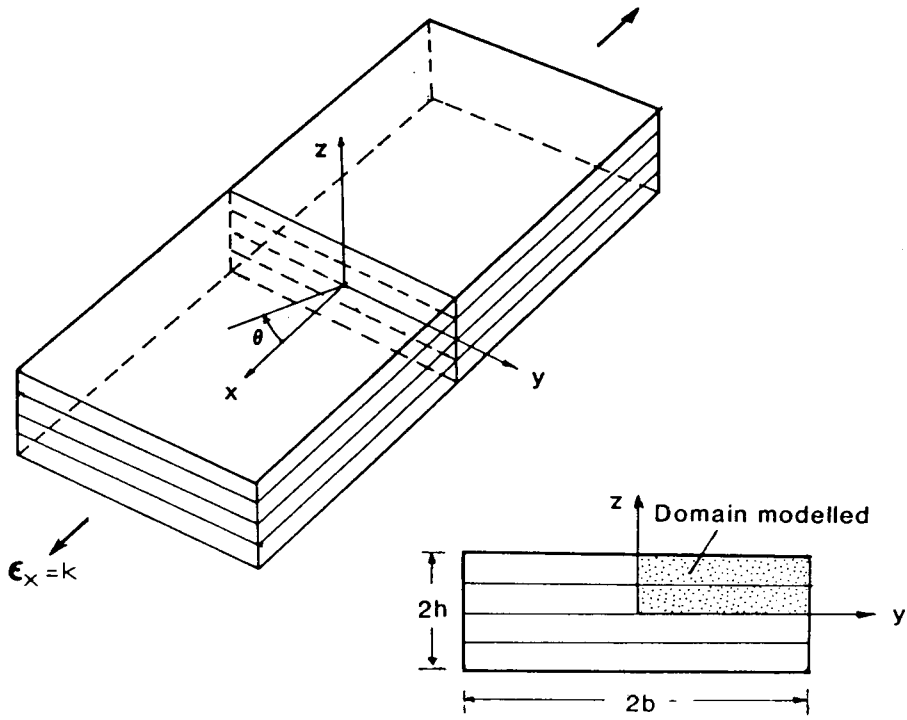


Figure 1. Laminate geometry and loading (cap is not shown).

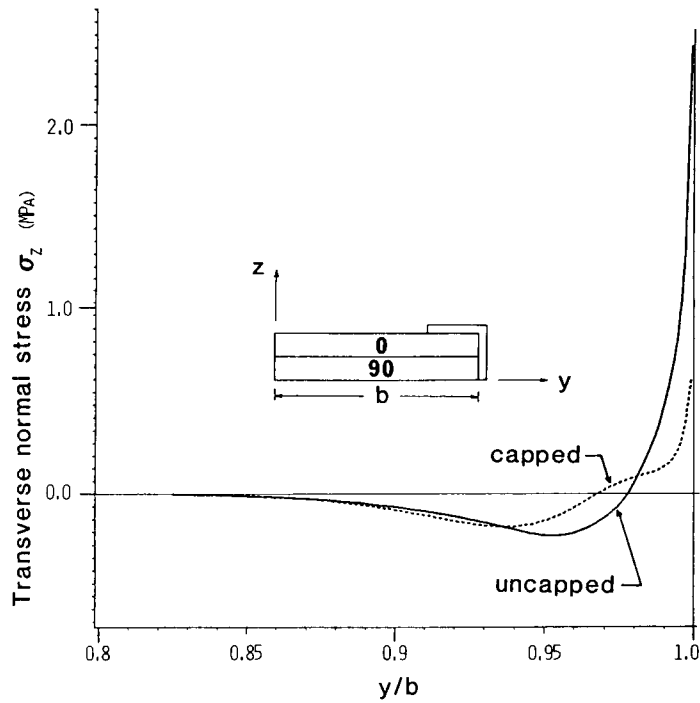


Figure 2. Distribution of the transverse normal stress along the width of a cross-ply laminate $[0^\circ/90^\circ]_5$.

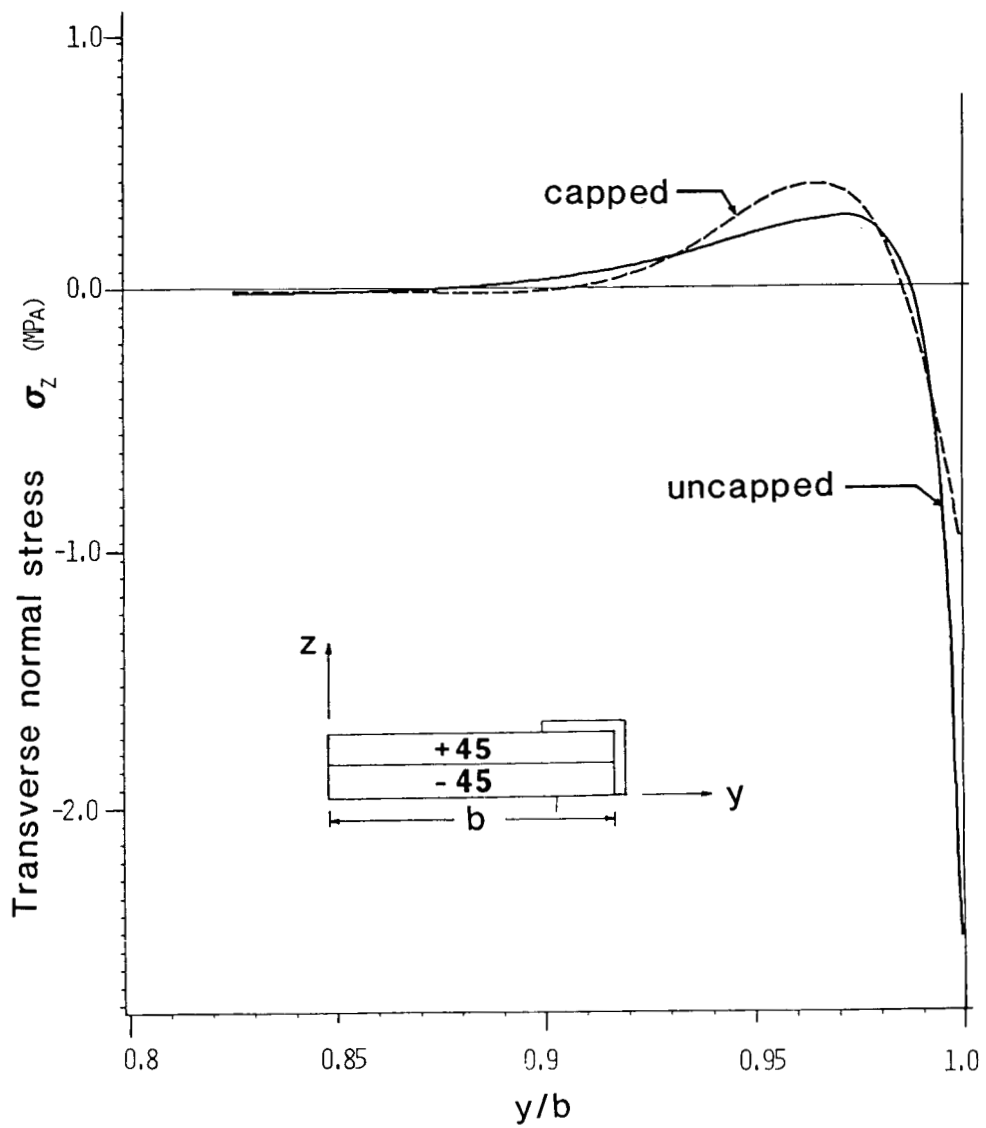


Figure 3. Distribution of the transverse normal stress (σ_z) along the width (y/b) of the laminate $[45^\circ/-45^\circ]_s$.

GLOBAL FUNCTIONS IN GLOBAL-LOCAL FINITE-ELEMENT ANALYSIS
OF
LOCALIZED STRESSES IN PRISMATIC STRUCTURES

Stanley B. Dong
University of California
Los Angeles, California

Abstract

An important consideration in the global local finite-element method (GLFEM) is the availability of global functions for the given problem. The role and mathematical requirements of these global functions in a GLFEM analysis of localized stress states in prismatic structures are discussed. A method is described for determining these global functions. Underlying this method are theorems due to Toupin and Knowles on strain energy decay rates, which are related to a quantitative expression of Saint-Venant's principle. It is mentioned that a mathematically complete set of global functions can be generated, so that any arbitrary interface condition between the finite element and global subregions can be represented. Convergence to the true behavior can be achieved with increasing global functions and finite-element degrees of freedom. Specific attention is devoted to mathematically two-dimensional and three-dimensional prismatic structures. Comments are offered on the GLFEM analysis of NASA flat panel with a discontinuous stiffener. Methods for determining global functions for other effects are also indicated, such as steady-state dynamics and bodies under initial stress.

Introduction

The finite-element method (FEM) has revolutionized structural and stress analyses in the last quarter of this century. Its impact has been widespread, even extending beyond the preserve of structural engineers to other fields. Although FEM is acknowledged as an extremely powerful modeling technique, the analysis community with its collective experience will admit that it is not the quintessential technique. There are problems not well suited to FEM that result in clumsy, ineffective and costly mathematical models. Examples can be cited from problems involving stress singularities and infinite domains. To obviate the difficulties, modifications to FEM have been explored. One alternate approach which bodes considerable promise is the so-called Global-Local Finite-Element Method (GLFEM).

GLFEM utilizes both conventional finite elements and classical Ritz functions in the modeling process. Their respective roles are readily apparent; finite elements work well in regions where complicated geometry and inhomogeneous material characterizations prevail, and Ritz functions, hereinafter referred to as global functions in GLFEM, enable the behavior other regions to be represented accurately and efficiently. At this stage of development, GLFEM can be assessed to be in its maturing phase. It is of good lineage, has already exhibited an enhanced capability above FEM in certain problems, and promises effectiveness in other classes of problems upon its full development.

Herein, GLFEM as applied to the analysis of localized stresses in prismatic structures is discussed. First, the essence of GLFEM and various GLFEM modeling layouts are summarized. A brief review of some problems that have been successfully analyzed by GLFEM is given. Then, the main theme relating to GLFEM analysis of localized stress states is addressed. Prismatic structures that can be described mathematically by two spatial variables are discussed first. Attention is devoted to the global functions, their development and their roles in the present setting. Then, three-dimensional structures are considered, with reference to the NASA example problem, where an outline of a method of attack is given. Last, comments on the analysis of localized stresses involving steady-state dynamic effects as well as other conditions are given.

Basic Concepts of GLFEM and the Various Mesh Configurations

Hamilton's principle, or alternatively the theorem of minimum potential energy when no inertial effects are present, may be considered as the basis for generating GLFEM equations. The theory and variational derivation of these equations may be found in Ref. [1, pp.451-474]. Also included therein is a survey of GLFEM contributions to the literature up to 1982.

As noted earlier, the technique utilizes finite-element modeling with classical Ritz approximations simultaneously. It enjoys the advantages of

more versatile modeling capabilities with substantially fewer degrees of freedom. Various global/local modeling configurations are illustrated in Fig. 1. Figs. 1a and 1f represent, respectively, the classical Ritz and finite-element configurations. The others are possible GLFEM mesh layouts. In a given problem, the modeling may take the form of any one of these configurations or a combination of two or more of them for various subregions. An important key is the enforcement of kinematic inter-regional continuity between various global and local subregions by means of constraint equations. In problems on localized stress states, only the Fig. 1c configuration will be used, where finite elements exclusively are used in one subregion and global functions in the other. Moreover, the global subregion may be infinite in extent.

The governing matrix equations in a GLFEM analysis have the form:

$$\begin{bmatrix} [K_{ll}] & [K_{lg}] \\ [K_{gl}] & [K_{gg}] \end{bmatrix} \begin{bmatrix} \{\delta\} \\ \{S\} \end{bmatrix} + \begin{bmatrix} [M_{ll}] & [M_{lg}] \\ [M_{gl}] & [M_{gg}] \end{bmatrix} \begin{bmatrix} \{\delta\} \\ \{S\} \end{bmatrix} = \begin{bmatrix} \{F_l\} \\ \{F_g\} \end{bmatrix} \quad (1)$$

where $\{\delta\}$ denotes the finite-element degrees of freedom and $\{S\}$ contains the array of generalized coordinates associated with the global functions. In Eq. (1), $[K_{gg}]$, $[M_{gg}]$, $[K_{ll}]$, and $[M_{ll}]$ refer to the global and local stiffness and mass matrices of the system. The matrices $[K_{lg}] = [K_{gl}]^T$ and $[M_{lg}] = [M_{gl}]^T$ represent global-local coupling from imposing kinematic continuity at interface(s) between subregion(s). Details on the formation of these matrices may be found in Ref. [1].

It is mentioned that GLFEM variants are possible, which do not lead to the same set of governing equations as Eq. (1). These variants contain the spirit of GLFEM and employ the modeling configurations shown in Fig. 1; however, the method of enforcing inter-regional continuity may differ. An application concerned with elastic wave scattering will illustrate one such variant.

Another key point in GLFEM is the availability of an appropriate set of global functions for a given problem or a class of problems. The accuracy and effectiveness of the method are dependent upon the quality of the global functions. The choice of these global functions for the analysis of localized stresses in prismatic structures and their method of derivation will be discussed in what follows. It will become apparent why these global functions, together with the finite-element model of the subregion that contains the localized stresses, will lead to a superior model.

Some Examples of Global Functions for GLFEM

Two areas ideally suited to GLFEM are fracture mechanics and infinite

and/or semi-infinite domain problems. Much has been published on various aspects of fracture mechanics problems. Many numerical methods have been used, many falling within a GLFEM classification or its variants. The global subregion model usually takes the form of special crack tip elements, where the singular stress field is incorporated into the stiffness matrix. These elements are well known. No further elaboration on this subject will be given here. Elastostatic analyses of half-space problems, wherein the far field behavior is represented by global functions (for example, Boussinesq or Cerruti solutions), have also met with considerable success. A number of references on both of these subjects may be found in Ref. [1].

Herein, two recent GLFEM applications are mentioned to emphasize the roles of the global functions and their mathematical suitability. They are concerned with (1) steady-state elastic wave scattering by axisymmetric objects embedded in an infinite isotropic medium and (2) steady-state soil-structure interaction involving an axisymmetric structure occupying some locale in a semi-infinite medium. The feature of note is that these global functions constitute a complete set of eigenfunctions and have the capability of mathematically representing an arbitrary scattered field to any given accuracy. Hence, the true behavior in the far field can be achieved.

In Fig. 2 is shown an elastic, axisymmetric inclusion embedded in an elastic, isotropic medium. Because finite elements are used for the object it may have inhomogeneous, orthotropic properties. The finite-element subregion includes this object and a portion of the surrounding medium. For convenience in the analysis, the interface is taken to be spherical. Outside of the finite-element subregion is the outer field, where a complete set of outgoing spherical harmonics is used to model the scattered field. Each component satisfies the equations of motion and the Sommerfeld radiation conditions. The global functions have specific stress and displacement distributions at the interface, and their undetermined strengths are the global function coefficients or the terms in $\{S\}$. A given incident wave illuminates this object. The scattered field is determined by solving the finite-element equations and requiring that the sum of incident and scattered wave fields based on the global functions have both traction and displacement continuity with the finite-element data at the interface. Details of this analysis may be found in Ref. [2]. Here, attention is called to the mathematical flexibility of the global functions for accommodating interface continuity to any precision with a sufficient number of terms.

The dynamic soil-structure interaction problem under steady-state conditions is shown in Fig. 3. The approach used here is similar to that for elastic wave scattering by an object embedded in the entire space. In fact, the same set of spherical harmonics for the entire space may be applied to this half-space problem. However, traction-free surface conditions are not satisfied by the spherical harmonics. Thus, in addition to traction and displacement continuity at the hemispherical interface, it is necessary to enforce the traction-free surface in the global subregion. In Refs. [3,4], details concerning an integral

constraint condition to meet this traction-free surface condition are given. Again, it is noted that because a complete set of global functions is used (that is, a set capable of modeling any arbitrary traction and displacement conditions between various subregions in a GLFEM layout), the analysis procedure enjoys the opportunity of converging onto the true behavior with increasing FEM and global degrees of freedom.

Mathematically Two-Dimensional Structures

The choice of global functions for mathematically two-dimensional structures will now be discussed. As illustrations of this class of problems and their GLFEM layouts, refer to Fig. 4, where examples of a laminated composite plate and cylinders are given. The double lap joint may be considered as a plane strain problem herein. The scarf joint joining two cylinders may be taken as an axisymmetric structure under axisymmetric or asymmetric loads. The purpose is to study the stresses in these joints.

Uniform stress states exist at points well away from these localized stress regions. If FEM were used, it is obvious that an awkward model would result. In GLFEM, two-dimensional finite elements (planar or axisymmetric toroidal elements) are used for the subregion containing the localized stresses. If the localized stress state contains a singularity, a global subregion within the finite-element subregion may be added. The interface location is dependent on the global functions' mathematical capability for capturing the transitional stress and displacement fields accurately. For global functions capable of representing the true behavior, the finite-element subregion can be quite small with the interface(s) near to the localized stress area. An independent set of global functions must be adopted at each interface. For the lap joint in Fig. 4, two or three distinct systems of global functions may be needed depending on the thickness and material properties of the plate components. Each set of global functions is associated with its own set of generalized coordinates or global coefficients. For the cylindrical scarf joint, two independent sets are needed.

The global functions in these cases are based on theorems relating to a quantitative expression of St. Venant's principle. Toupin [5] and Knowles [6] presented upper bound estimates of strain energy decay rates in terms of distance from a self-equilibrated stress state. Their results can be stated in the form of a strain energy inequality:

$$V(x) \leq V(0) e^{-2\gamma x} \quad (2)$$

where γ is the inverse of the characteristic decay length, $V(0)$ is the total strain energy and $V(x)$ is that portion of $V(0)$ in the body beyond x . Since the strain energy is quadratic, the mechanical variables such as stress, strain and displacement are of the forms:

$$u_j(x) \leq K_1 e^{-\gamma x} \quad \epsilon_{ij}(x) \leq K_2 e^{-\gamma x} \quad \sigma_{ij}(x) \leq K_3 e^{-\gamma x} \quad (3)$$

where K_i ($i=1,2,3$) are constants.

Based on these theorems, a boundary-value problem can be formulated for a prismatic structure. Using the solution form in the prismatic direction as $e^{-\gamma x}$, the analysis leads to an eigenvalue problem. The eigenvalues γ 's are the characteristic inverse decay lengths and the eigenfunctions are distributions of self-equilibrated stress states. These eigendata comprise a complete set from which any arbitrary self-equilibrated stress state may be represented. These eigendata may be used as global functions for describing the far-field behavior in a prismatic structure. Horgan and his colleagues have solved a number of problems on homogeneous and sandwich plates under plane strain using the Airy stress function as the primary dependent variable (see, for examples, Refs. [7,8]).

For a laminated composite structure, it is more convenient to determine the eigendata numerically. Dong and Goetschel [9] developed a one-dimensional finite-element analysis for extracting eigendata for a laminated composite plate with an arbitrary number of bonded, elastic laminates. Finite-element discretization occurs in the thickness direction, see Fig. 5. Applying the theorem of minimum potential energy, a system of second-order ordinary differential equations is obtained. By invoking exponential decay $e^{-\gamma x}$, the following second-order algebraic eigenvalue problem results:

$$[K_1]\{Q\} - \gamma[K_2]\{Q\} + \gamma^2[K_3]\{Q\} = 0 \quad (4)$$

where $\{Q\}$ is an ordered set of the plate's nodal displacements. This equation is reducible to first order with a non-symmetric matrix. If a large number of degrees of freedom are involved, a Block-Stodola iteration technique [10] can be used to extract the eigendata efficiently. The solution consists of a complete set of eigenvalues and corresponding eigenvectors, which are the self-equilibrated displacement states for the given composite plate. Stresses can be computed from these displacements.

Laminated cylinders may also be solved using the same finite-element scheme, see Ref. [11]. The mechanical variables have circumferential dependence, which may be expressed analytically by Fourier series. As a circumferential mode number m occurs in this case, the counterpart to Eq. (4) for each circumferential mode has the form:

$$[K_1(m)]\{Q\} - \gamma[K_2(m)]\{Q\} + \gamma^2[K_3(m)]\{Q\} = 0 \quad (5)$$

The solution to Eq. (4) or (5) provides the global function data base for the numerical evaluation of the global stiffness matrix and the

global-local coupling as a prelude to a mathematically two-dimensional GLFEM analysis of the localized stress zones. Some preliminary results of this type have been obtained, which are contained in Refs. [12,13]. These limited scope studies indicate an overall feasibility for this approach.

The accuracy of the global functions depends on the fineness of the one-dimensional finite-element model adopted for the eigenproblem. Since only one-dimensional finite elements are used, a large model does not incur an inordinate computational effort because of a very small bandwidth.

The number of global functions required in a GLFEM analysis depends on both the nature of the localized stress and the location of the interface. Having an interface near the localized stress zone will require a larger number of global functions, but with a decrease in the finite-element coordinates. Conversely, an interface far removed from the localized zone needs fewer global functions, but is counteracted by a greater number of finite-element degrees of freedom.

Three-Dimensional Structures and the NASA Problem

A schematic of a three-dimensional prismatic structure and the NASA problem of a flat stiffened composite panel with a discontinuous stiffener are shown in Fig. 6. In this class of problems, three-dimensional finite elements must be employed in the localized stress region. Global functions must be used at the interface. They can be obtained from a two-dimensional finite-element analysis of the inverse characteristic decay lengths.

The analysis to determine the global functions follows the same methodology as that for mathematically two-dimensional structures. The prismatic cross-section is modeled by two-dimensional finite elements. With the dependence in the prismatic direction taken as e^{-YX} , an eigenproblem emerges for the extraction of eigendata that form the global function data base for the given cross section. The other aspects are the same as that described in the previous section. It is obvious that, in this case, the computational effort is greater.

Some comments can be given on a GLFEM analysis of the NASA flat panel. The set of two-dimensional global functions constitutes a complete system of eigenfunctions, with the non-zero eigenvalues associated with inverse characteristic decay lengths of self-equilibrated stress states. There are two zero eigenvalues for two stress distributions exhibiting no decay. They are the uniform axial deformation and pure bending states. These two global functions are needed in a GLFEM analysis of the NASA flat panel, since the discontinuous stiffener may produce bending in addition to its uniform end shortening. A set of global functions with all of these members present should permit a three-dimensional finite-element model to be concentrated on the details of the discontinuous stiffener region. Parametric studies wherein the hole and the gap length in the discontinuous stiffener are varied may be conducted. Each configuration will require a change of the three-dimensional finite-element mesh, but the same set of global functions may be used in all cases.

Applications to Steady-State Problems

The discussion of global functions in the two earlier sections pertained to elastostatic analysis of the localized stress zones. Here some remarks on steady-state dynamic effects are made. The one-dimensional finite-element method for generating global function data bases can be modified for steady-state inertial effects by including kinetic energy in the problem formulation. Instead of Eqs. (4) and (5), those equations become, respectively:

$$[K_1]\{Q\} - \gamma[K_2]\{Q\} + \gamma^2[K_3]\{Q\} + \omega^2[M]\{Q\} = 0 \quad (6)$$

$$[K_1(m)]\{Q\} - \gamma[K_2(m)]\{Q\} + \gamma^2[K_3(m)]\{Q\} + \omega^2[M]\{Q\} = 0 \quad (7)$$

where ω is the steady-state forcing frequency. The derivations of these equations are given in Refs. [11,14].

With these global functions, it is possible to study elastic wave scattering in prismatic structures by discontinuities during vibration or by some other steady-state dynamic input. GLFEM analysis of this type of prismatic structures will be similar to problems of elastic wave scattering by an object embedded in an infinite medium or soil-structure interaction.

Effects of Initial Stress

Using the same methodology, prismatic structures under initial stress may also be analyzed. In this case, the global functions must include the prestressing effect. One-dimensional finite-element analysis of wave propagation in laminated composite plates and cylinders under initial stress have been explored, see Refs. [15,16]. It is a straightforward task to adapt these formulations to generate an eigenproblem for the global functions for a prismatic structure under initial stress. Also, no conceptual difficulties are seen in an extension to three-dimensional prismatic structures under prestress.

Concluding Remarks

Considerable discussion has been devoted to the strategies of GLFEM analyses of prismatic structures with localized stress regions and other discontinuities. The role of the global functions has been clearly outlined and their mathematical requirements indicated. The method for deriving these global functions for prismatic structures, whose cross-sectional geometries are complicated by laminated construction, has been discussed. From the discussion of GLFEM analysis strategy, it should be clear that GLFEM is feasible and effective. Considerable economy of computational efforts over a strictly FEM approach should be realized.

References

1. Dong, S.B., "Global-Local Finite Element Methods," State-of-the-Art Surveys on Finite Element Technology, edited by A.K. Noor and W.D. Pilkey, Chap. 14, pp.451-474, American Society of Mechanical Engineers, 1983.
2. Goetschel, D.B., Dong, S.B. and Muki, R., "A Global-Local Finite Element Analysis of Axisymmetric Scattering of Elastic Waves," Journal of Applied Mechanics, American Society of Mechanical Engineers, Vol. 49, pp. 816-820, 1982.
3. Avanesian, V., Muki, R. and Dong, S.B., "Forced Oscillation of an Axisymmetric Structure in Contact with an Elastic Half-Space by a Version of Global Local Finite Element," Journal of Sound and Vibration, Vol. 104, pp. 449-463, 1986.
4. Avanesian, V., Muki, R. and Dong, S.B., "Axisymmetric Soil-Structure Interaction by Global-Local Finite Elements," International Journal of Earthquake Engineering and Structural Dynamics, in press.
5. Toupin, R.A., "Saint-Venant's Principle," Archive for Rational Mechanics and Analysis, Vol. 18, pp. 83-96, 1965.
6. Knowles, J.K., "On Saint-Venant's Principle in Two-Dimensional Linear Theory of Elasticity," Archive for Rational Mechanics and Analysis, Vol. 21, pp. 1-22, 1966.
7. Choi, I. and Horgan, C.O., "Saint-Venant's Principle and End Effects in Anisotropic Elasticity," Journal of Applied Mechanics, American Society of Mechanical Engineers, Vol. 44, pp. 424-430, 1977.
8. Horgan, C.O., "Some Remarks on Saint-Venant's Principle for Transversely Isotropic Composites," Journal of Elasticity, Vol. 2, pp. 335-339, 1972.
9. Dong, S.B. and Goetschel, D.B., "Edge Effects in Laminated Composite Plates," Journal of Applied Mechanics, American Society of Mechanical Engineers, Vol. 49, pp. 129-135, 1982.
10. Dong, S.B., "A Block-Stodola Eigensolution Technique for Large Algebraic Systems with Non-Symmetrical Matrices," International Journal for Numerical Methods in Engineering, Vol. 11, pp. 247-267, 1977.
11. Huang, K.H. and Dong, S.B., "Propagating Waves and Standing Vibrations in a Composite Cylinder," Journal of Sound and Vibration, Vol. 96, pp. 363-379, 1984.
12. Chen, W.T., "Studies Relating to Application of the Global-Local Finite Element Method to Elastostatic Analysis of End Effects in Cylinders," M.S. Research Project Report, Dept. of Civil Engineering, University of California, Los Angeles, March 1985.

13. Nystrom, D.M., "Numerical Evaluation of Global Stiffness Matrix for Analysis of Self-Equilibrated Stress States," M.S. Thesis, Graduate Division, University of California, Los Angeles, June 1985.
14. Dong, S.B. and Huang, K.H., "Edge Vibrations in Laminated Composite Plates," Journal of Applied Mechanics, American Society of Mechanical Engineers, Vol. 52, pp. 433-438, 1985.
15. Bradford, L.G. and Dong, S.B., "Natural Vibrations of Orthotropic Plates under Initial Stress," International Journal of Solids and Structures, Vol. 11, pp. 213-230, 1975.
16. Bradford, L.G. and Dong, S.B., "Natural Vibrations of Orthotropic Cylinders under Initial Stress," Journal of Sound and Vibration, Vol. 60, pp. 157-175, 1978.

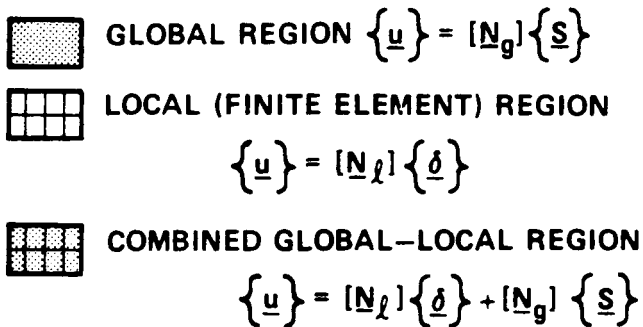
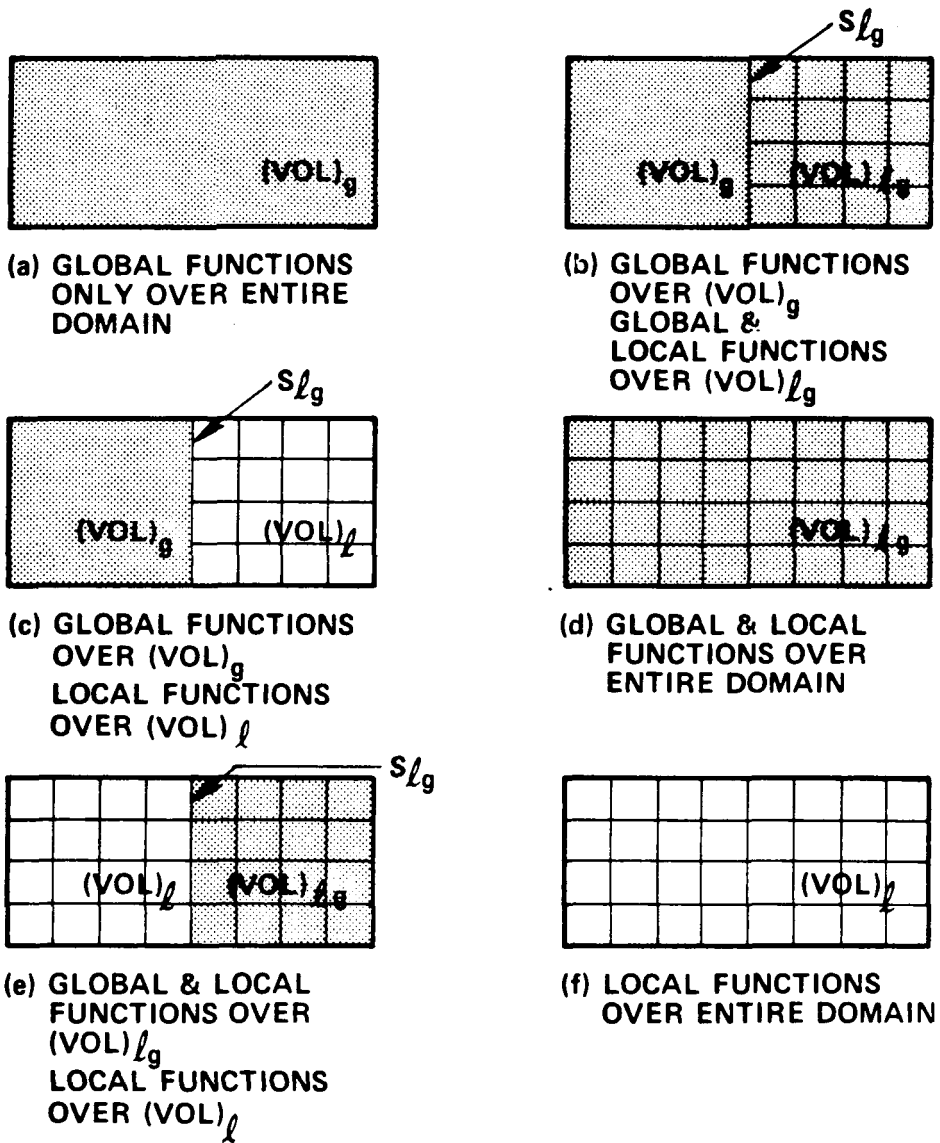
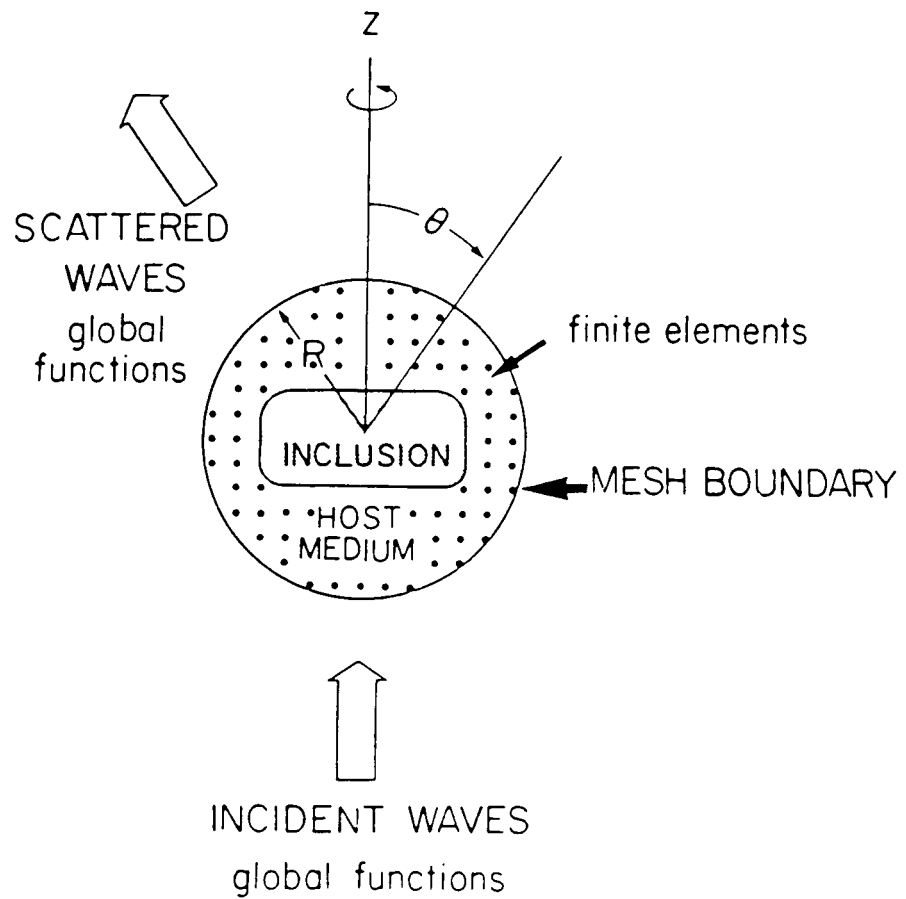


Figure 1. Basic global-local mesh configurations.



SCATTERED WAVE FIELD - SPHERICAL HARMONICS

$$u_r(\theta) = \sum_{n=0}^N \left[A_n D_{1n} - B_n S_{1n} \right] P_n(\cos\theta)$$

$$u_\theta(\theta) = \sum_{n=0}^N \left[A_n D_{2n} - B_n S_{2n} \right] \frac{dP_n(\cos\theta)}{d\theta}$$

$$\sigma_{rr}(\theta) = \sum_{n=0}^N \left[A_n D_{3n} - B_n S_{3n} \right] P_n(\cos\theta)$$

$$\sigma_{r\theta}(\theta) = \sum_{n=0}^N \left[A_n D_{4n} - B_n S_{4n} \right] \frac{dP_n(\cos\theta)}{d\theta}$$

Figure 2. Elastic wave scattering by axisymmetric inclusion embedded in an infinite isotropic medium.

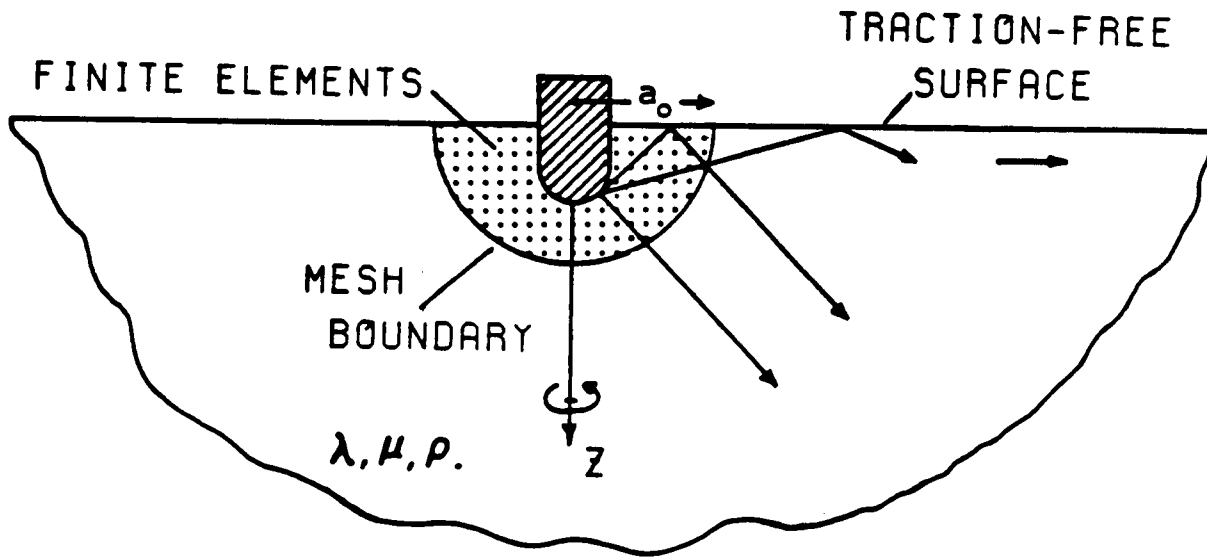


Figure 3. Soil-structure interaction problem.

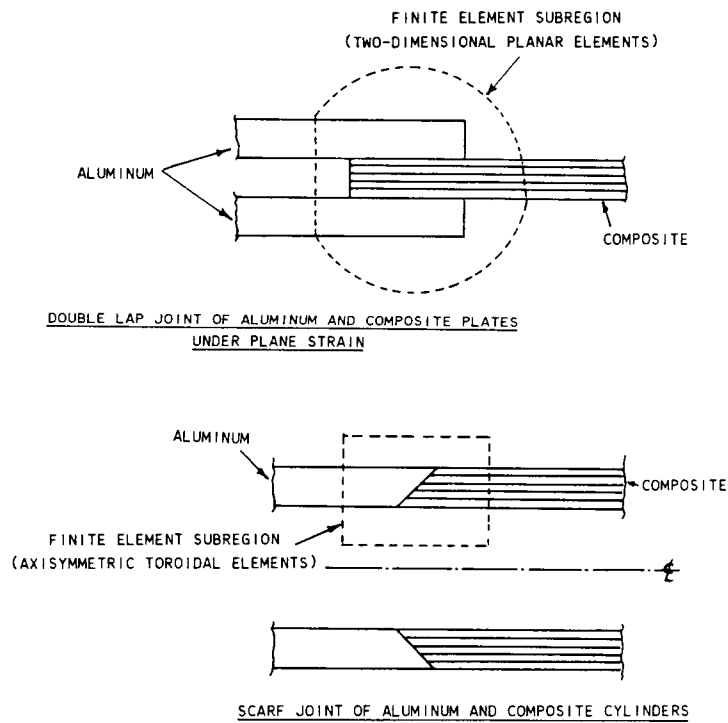
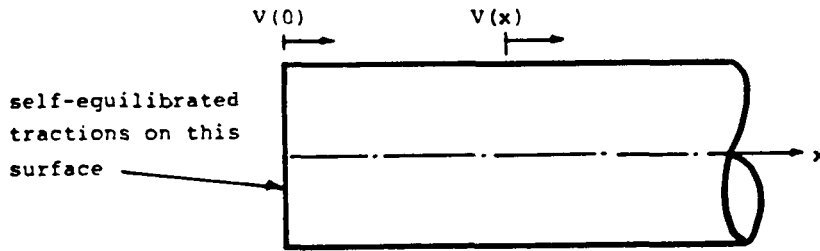


Figure 4. Two-dimensional prismatic structures.

STRAIN ENERGY DECAY RATE

$$V(x) \leq V(0) \cdot \exp(-2\gamma x)$$

$$\begin{aligned} u_i(x) &\leq K_1 \cdot \exp(-\gamma x) \\ \epsilon_{ij}(x) &\leq K_2 \cdot \exp(-\gamma x) \\ \tau_{ij}(x) &\leq K_3 \cdot \exp(-\gamma x) \end{aligned}$$

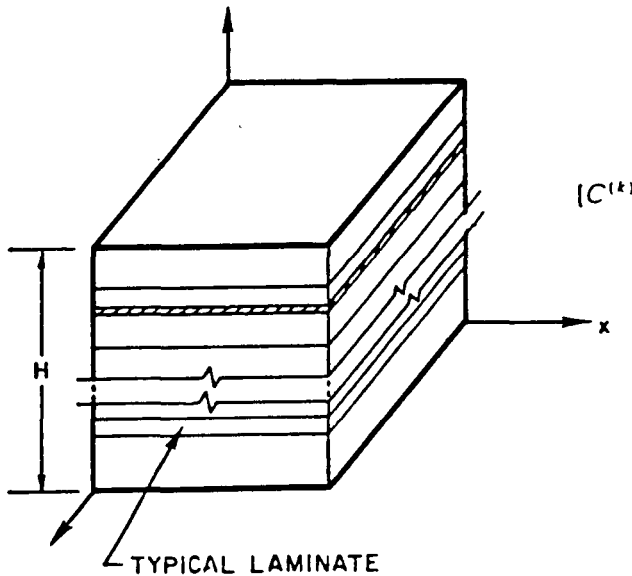


FORMULATION OF TWO-DIMENSIONAL PROBLEM FOR PLATE WITH PLANE ANISOTROPIC MATERIALS

$$\begin{aligned} \Pi = \frac{1}{2} \int \left\{ \sum_k \int |\epsilon|^T [C^{(k)}] |\epsilon| dy \right\} dx \\ - \int \left\{ \tau_{xx} u(0,y) + \tau_{xy} v(0,y) \right\} dy \end{aligned}$$

WHERE

$$\epsilon_{xx} = u_{,x}; \quad \epsilon_{yy} = v_{,y}; \quad \gamma_{xy} = u_{,y} + v_{,x}$$



DISPLACEMENT FIELD

$$\begin{aligned} u(x,y) &= n_1(y) \cdot u_b(x) + n_2(y) \cdot u_m(x) + n_3(y) \cdot u_f(x) \\ v(x,y) &= n_1(y) \cdot v_b(x) + n_2(y) \cdot v_m(x) + n_3(y) \cdot v_f(x) \end{aligned}$$

INTERPOLATION FUNCTIONS (QUADRATIC POLYNOMIALS)

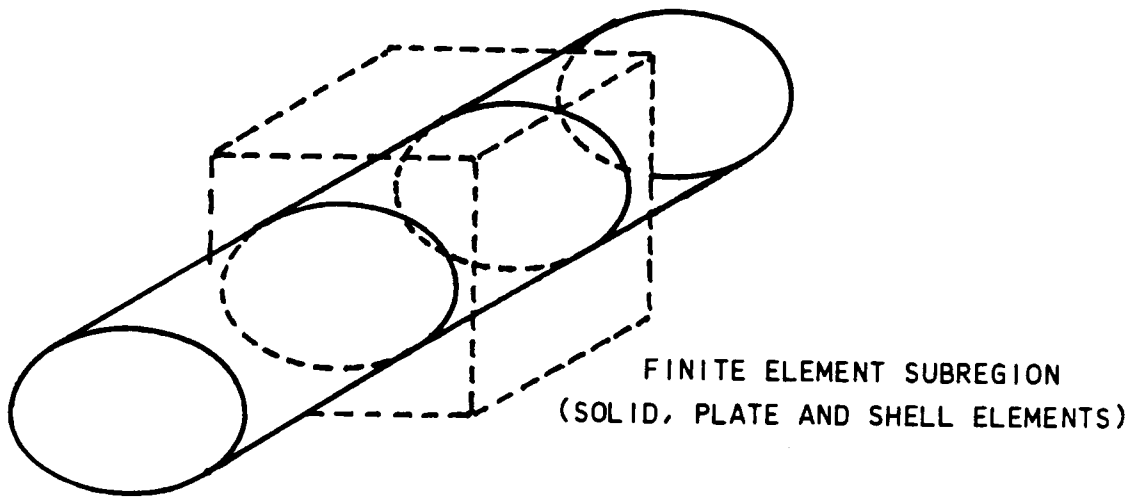
$$\begin{aligned} n_1 &= 1 - 3\zeta + 2\zeta^2; \quad n_2 = 4\zeta - 4\zeta^2; \quad n_3 = 2\zeta^2 - \zeta \\ \zeta &= (y - y_b) / (y_f - y_b) \end{aligned}$$

SOLUTION FORM IN X-DIRECTION SECOND ORDER ALGEBRAIC EIGENVALUE PROB.

$$|Q| = |Q_0| \cdot \exp(-\gamma x)$$

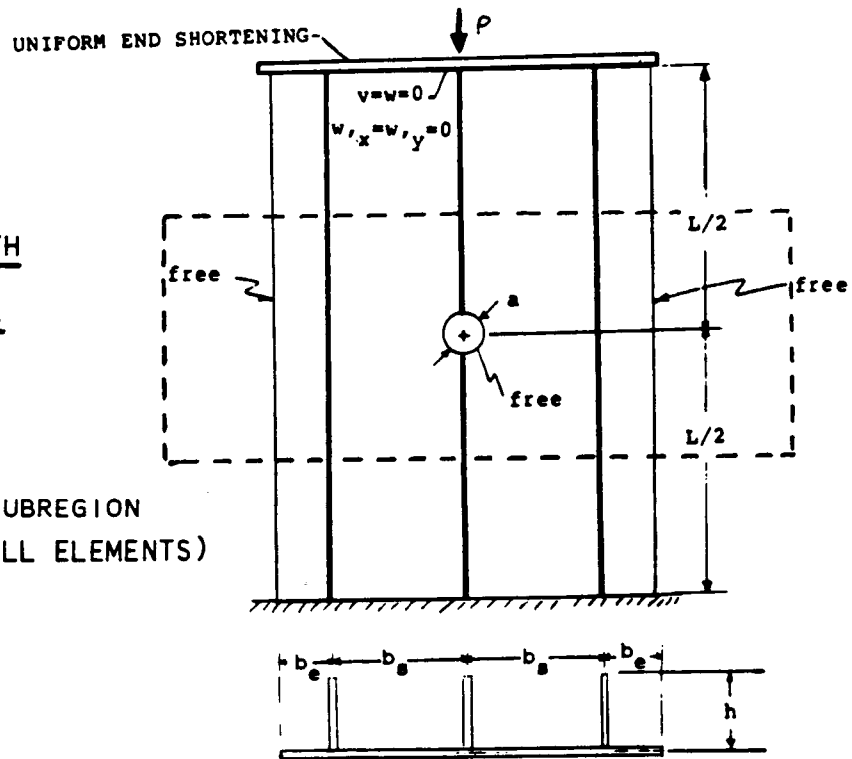
$$[\gamma^2 [K_3] + \gamma [K_2] + [K_1]] |Q_0| = 0$$

Figure 5. Finite-element analysis of self-equilibrated edge effects in a composite plate.



EXAMPLE OF A
THREE-DIMENSIONAL PRISMATIC STRUCTURE

THE NASA FLAT PANEL WITH
DISCONTINUOUS STIFFENER



FINITE ELEMENT SUBREGION
(SOLID, PLATE AND SHELL ELEMENTS)

Figure 6. Three-dimensional prismatic structures.

GLOBAL-LOCAL METHODOLOGIES AND THEIR APPLICATION
TO NONLINEAR ANALYSIS

Ahmed K. Noor
George Washington University
NASA Langley Research Center
Hampton, Virginia

ABSTRACT

An assessment is made of the potential of different global-local analysis strategies for predicting the nonlinear and postbuckling responses of structures. Two postbuckling problems of composite panels are used as benchmarks and the application of different global-local methodologies to these benchmarks is outlined. The key elements of each of the global-local strategies are discussed and future research areas needed to realize the full potential of global-local methodologies are identified.

NOMENCLATURE

E_L, E_T	Elastic moduli of the individual layers in the direction of fibers and normal to it, respectively
G_{LT}, G_{TT}	Shear moduli in plane of fibers and normal to it, respectively
h	Thickness of panel
L_1, L_2	Side lengths of panel
N	Total axial force acting on the edge of the panel
q	Edge displacement
R	Radius of curvature of the panel middle surface
r	Number of global approximation vectors
U	Total strain energy of the panel
u_α, w	Displacement components in the coordinate directions
x_α, x_3	Orthogonal curvilinear coordinate system
ν_{LT}	Major Poisson's ratio of the individual layers
ϵ_1	Axial strain
The range of the subscript α is 1,2.	

1. INTRODUCTION

Considerable progress has recently been made in computational mechanics which is manifested by the development of versatile and powerful finite-element discretization methods, improved numerical algorithms and programming techniques (see, for example, BELYTSCHKO & HUGHES [1983]; NOOR & PILKEY [1983]; NOOR [1983]; LIU, BELYTSCHKO & PARK [1984]; and KARDESTUNCER [1985]). Also, an explosive growth has taken place in computer technology. In particular, the introduction of large expensive computer systems, usually referred to as supersystems, such as CRAY X-MP, CDC CYBER 205 and Denelcor HEP1 has made possible new levels of sophistication in the modeling of complex structures which were not possible before (NOOR, STORAASLI & FULTON [1984]). In spite of these advances the detailed stress analysis of complex structures is very time consuming and, therefore, is not economically feasible. To date the only realistic structural response simulations that have been obtained

involve either simple structural configurations or components of complicated configurations. Prediction of the response of future complex structures such as those of flight vehicles is likely to require more sophisticated analysis models than has heretofore been done. This is because of the requirements of high performance, light weight and economy and the associated stringent design criteria. Also, analysis may of necessity replace tests in some mission-critical areas.

Among the different analysis methodologies that have high potential for the accurate prediction of detailed stress distribution in structures without overtaxing the available computational resources are the global-local methodologies which are basically hybrid modeling and/or analysis techniques. In order to put these methodologies in proper perspective, a brief summary is given of the different approaches for reducing the cost and/or time for solving nonlinear problems. The efforts devoted to this activity can be grouped into three different levels.

The first level is modeling. Reductions in cost of analysis can be achieved by using simple models that capture major effects in the responses and by exploiting all the symmetries and quasi-symmetries in the problem (see NOOR & PETERS [1985]; and NOOR, ANDERSEN & TANNER [1985]).

The second level is that of computational strategies. Significant reductions in time can be achieved by incorporating the known physical behavior into the computational model of the structure and by using global-local methodologies in which different analysis methods and/or models are coupled for predicting the nonlinear response of the structure.

The third level is that of numerical algorithms. These include fast algorithms for solution of equations (e.g., multigrid methods, operator splitting techniques, dynamic relaxation, and element-by-element techniques -HACKBUSCH & TROTTENBERG [1982]; UNDERWOOD [1983]; and HUGHES, RAEFSKY, MULLER, WINGET & LEVIT [1984]); as well as the vectorized and parallel numerical algorithms for use on pipeline and parallel processors (SCHENDEL [1984]; MIKLOSKO & KOTOV [1984]; and PADDON [1984]).

The present study deals with global-local methodologies which belong to the second category. Specifically, the objectives of this paper are:

- 1) To review and assess the potential of a number of different global-local analysis strategies for predicting the nonlinear and postbuckling responses of structures
- 2) To identify the future directions for research required to realize their full potential

Discussion of global-local methodologies is primarily focused on nonlinear analysis of composite panels with discontinuities (e.g., stiffeners and cutouts). Two benchmark problems of composite cylindrical panels with cutouts typical of those used in modern aircraft structures are selected to provide a focus for the discussion. However, many of the conclusions apply to other complex structural configurations.

The paper is divided into three parts. The first part describes the two benchmark problems, identifies their major characteristics, and lists the difficulties encountered in analyzing them using conventional finite-element methods. In the second part of the paper four global-local analysis strategies are reviewed and the potential for using these strategies in analyzing the benchmark problems is assessed.

The third part of the paper identifies the items that pace the progress of global-local methodologies and their application to nonlinear analysis. These are the recommended future directions of research.

2. BENCHMARK PROBLEMS AND THEIR CHARACTERISTICS

The two benchmark problems selected for this study are shown in Fig. 1. They are postbuckling problems of laminated composite cylindrical panels with central circular cutouts. The loading consists of applied axial end displacements. One of the panels is unstiffened and the other has discrete blade stiffeners. The two panels are generic examples of modern composite aircraft components for which postbuckling strength is desired in the presence of local discontinuities such as holes and cracked stiffeners. In the conventional finite-element approach the panels are modeled using two-dimensional shell elements and the stiffeners are modeled using two-dimensional plate elements. The unstiffened panel was analyzed using an in-house research program. Shear-flexible mixed finite elements were used in the modeling (see NOOR & ANDERSEN [1982]). Also, extensive numerical solutions to this problem using continuum-based shell elements are presented in STANLEY [1985] in which the imperfection sensitivity of the panel is assessed. The stiffened panel was analyzed using the EISI-EAL program and the analysis results are given in KNIGHT, GREENE & STROUD [1985]. The characteristics of the finite-element models used in the present study and those used in KNIGHT, GREENE & STROUD [1985] are summarized in Table 1. The response of both panels exhibits inversion symmetry, and therefore, only one half of each panel needs to be analyzed (see NOOR, MATHERS & ANDERSON [1977]). The numbers between parentheses in Table 1 refer to the number of elements and degrees of freedom required for analyzing the full panel.

The response of the unstiffened panel is shown in Figs. 2 and 3. The postbuckling response of the panel exhibits a sudden drop in the loading due to delamination in the neighborhood of the cutout (see KNIGHT & STARNES [1984]). Figure 2 shows plots of the total axial load versus axial and normal displacements up to the maximum load reached. For the range of loading considered, the results shown in Fig. 2 agree reasonably well with the experimental and numerical results presented in KNIGHT & STARNES [1984].

Figure 3 shows normalized contour plots of the axial and normal displacements as well as the axial strains on the top and bottom surfaces. Note the high strain concentration at the cutout. A finer model near the cutout is required for the accurate prediction of the strain in that region. On the basis of the studies made, the following characteristics of the two benchmark problems can be identified:

1. The presence of discontinuities (cutouts and stiffeners) results in large numbers of degrees of freedom in the finite-element models.
2. The detailed stress analysis (including determination of interlaminar stresses) near the cutout requires either a higher order two-dimensional theory or a three-dimensional theory. Failure analysis (including prediction of delamination) requires even more sophistication in the modeling and analysis.
3. The postbuckling response exhibits large rotations in certain regions.
4. The postbuckling response of the unstiffened panel is highly sensitive to initial imperfections. Tracing the postbuckling response past the maximum load point requires the inclusion of initial imperfections in the model (see STANLEY [1985]).

3. GLOBAL-LOCAL ANALYSIS STRATEGIES

In this section, the application of four different global-local analysis

strategies to the prediction of the nonlinear response of benchmark composite panels is discussed. In each case, the key features of the global-local approach and the requirements for its effective implementation are identified.

3.1 Zooming Technique

The first global-local approach considered is the zooming technique in which a global solution is obtained using a coarse grid. Then the detailed stress distribution near the cutout (local solution) is obtained by zooming on that area, refining the model, and using the displacements from the coarser model as input for the refined model (see Fig. 4). Single or multiple levels of zooming can be made corresponding to multiple passes from coarse to fine subdivisions (WILKINS [1983], and HIRAI, UCHIYAMA, MIZUTA & PILKEY [1985]). Interelement compatibility can be maintained by either embedding the region of the fine grid into a superelement or by using transition elements at the interface between the coarse and the fine grids (ARMEN, Grumman Aircraft Systems, Bethpage, NY, Private Communication, 1985).

A number of questions remain in connection with this technique, namely:

1. Limitations of the technique for nonlinear problems
2. Criteria for selecting the extent of the region for refining the model and selection of the refined model
3. Treatment of the interfaces between the coarse and the fine grids and the effect of the error on the boundary data for the refined model on the accuracy of the stresses in that model

The early work on zooming techniques was based on heuristic and intuitive approaches for selecting the fine model. More recent work is based on the distribution of the strain energy density function. Currently, a variety of adaptive refinement techniques are available. Some of these techniques will be discussed in subsequent sections.

3.2 Simultaneous Application of Two Discretization Techniques

The early applications of this approach consisted of the simultaneous use of the finite-element method and the global (classical) variational technique (see MOTE [1971]). A variety of options are available depending on the extent of using each of the two techniques in the model (DONG [1983]). The concept has been later generalized to cover other techniques for the global (approximate) solution and the local (detailed) solution. A list of the common techniques used for generating these solutions is shown in Table 2.

The global solution can be obtained by using classical variational methods (e.g., Rayleigh-Ritz or weighted-residual approaches), or any of the discrete element methods (conventional finite elements, global elements, boundary element method) or their combinations. In the global (or macro) element method the structure is divided into a small number of elements and a suitable (usually nonpolynomial) approximation is made within each element. The continuity of the field variables across the interfaces is imposed implicitly in the variational functional (DELVES & HALL [1979], and DELVES & PHILLIPS [1980]). The global element method is a compromise between classical variational techniques and finite-element method. It combines the rapid convergence of global variational methods with the ability to handle complicated geometries.

The boundary element method is an effective technique for solution of linear and materially nonlinear structural problems with high stress gradients (or singularities). However, it is not competitive with either the finite-

element or the global element method for geometrically nonlinear problems (see KAMIYA & SAWAKI [1982]; and KAMIYA, SAWAKI & NAKAMURA [1984]).

Among the different techniques for local analysis are the discrete element methods and analytical solutions (e.g., polar and/or edge functions PATTIBIRAMAN, RAMAMURTI & REDDY [1974]). Figure 5 shows three different combinations of discrete element methods for analyzing the unstiffened composite panel.

The effective implementation of this global-local strategy requires:

- a) criteria for selecting the proper global and local analysis techniques and
- b) problem-adaptive strategies for generating global solutions and treatment of interfaces.

3.3 Reduction Methods

These are hybrid two-step techniques which are based on the successive application of a discrete element method (finite elements, boundary elements or combination of finite elements and boundary elements) and classical variational techniques (see, for example, NOOR [1982]; and NOOR & PETERS [1983]). The discrete element method is used to generate few global approximation vectors (or modes). The classical variational technique is then used to compute the amplitudes of these modes. The primary objective of using reduction methods is to reduce considerably the number of degrees of freedom in the initial discretization, and hence, reduce the computational effort involved in the solution of the nonlinear problem.

The application of reduction methods to the unstiffened composite panel problem is depicted in Figs. 6 and 7. Figure 6 shows the accuracy of the normal displacements and total strain energy obtained by using four global approximation vectors (generated at zero loading). Figure 7 shows contours of the normal displacement w of the first three global approximation vectors.

Two recent applications of reduction methods deserve further examination. In the first, only partial reduction is made. The degrees of freedom in the region of strong nonlinearity (e.g., near the cutout) are retained; the other degrees of freedom are reduced. In the second application, the response of a complex structure (e.g., stiffened panel) is generated using small (or large) perturbations from the response of a simpler system (e.g., unstiffened panel). It is also possible to use a hierarchy of simpler structural systems in generating the response of the original complex structure. This is accomplished by choosing a number of perturbation parameters, and successively applying a single-parameter reduction method with each of the parameters. Application of this strategy to the nonlinear analysis of anisotropic panels is described in NOOR [1985].

The wide acceptance of reduction methods and their incorporation into commercial programs requires: a) the selection of a simple set of global approximation vectors and b) the development of a problem-adaptive strategy for error sensing and control.

3.4 Hierarchy of Mathematical Models and/or Numerical Approximation Techniques

The last global-local approach considered is that based on a hierarchy of mathematical models for different parts of the structure. The application of this approach to the stiffened composite panel is depicted in Fig. 8 where a heuristic choice is made of the mathematical models. For the panel a boundary-layer (or a higher order two-dimensional) theory is used near the

cutout, followed by a first-order shear deformation theory, and then a classical shell theory. For the stiffeners, a plate theory is used near the cutout, followed by a thin-walled beam theory and then a shear deformation or a classical beam theory. The effective implementation of this approach requires the following:

1. Systematic procedure for generating the hierarchy of mathematical models (e.g., the method of initial functions of V. Z. Vlasov - VLASOV & LEONTEV [1966]; and IYENGAR, CHANDRASHEKHARAN & SEBASTIAN [1974]); or the asymptotic integration technique - GOLDENVEIZER [1976])
2. Criteria for the adaptive refinement of the mathematical model
3. Treatment of the interfaces between the different regions

4. TREATMENT OF INTERFACES

The treatment of interfaces is one of the key elements of the global-local analysis. The two commonly used approaches for maintaining displacement compatibility and traction reciprocity at the interfaces are: 1) Lagrange multiplier method; and 2) penalty function method. The second approach has the advantage that it does not lead to any extra unknowns or equations (DELVES & HALL [1979]). The numerical problem associated with increasing the penalty weight, to meet constraint satisfaction tolerances can be overcome by using the iterative procedure described in FELIPPA [1978].

5. QUALITY CONTROL OF NUMERICAL SOLUTIONS

One of the most difficult aspects of numerical modeling is the validation of the results and ensuring that a given model is adequate for the particular problem at hand. In general, there are three types of errors in the numerical solution. These errors are (see UTKU & MELOSH [1984]):

1. Mathematical modeling errors, which result from the simplifications made in abstracting the mathematical model from the real structure.
2. Discretization errors, which are caused by the numerical discretization of the continuous mathematical model.
3. Manipulation errors, which are caused by: a) the finite precision of the computers (limitation in representing real numbers due to the finiteness of the computer word length); and b) the errors resulting in the process of solving the equations of the discrete model (e.g., using iterative methods).

In this paper only the second type, namely: discretization errors, is considered. There are two classical approaches for estimating these errors (KELLY, GAGO & ZIENKIEWICZ [1983]).

1. Extension methods - based on reanalysis of the structure on a sequence of meshes of increasing refinements (h extension); with a hierarchic set of interpolation polynomials (p extension); or using a combination of the two (h-p extension).

2. Dual (or complementary) procedure - based on obtaining two solutions with two different computer programs to provide bounds on global response characteristics.

Both of these approaches are too expensive for practical implementation.

In recent years, considerable effort has been devoted to the development of a posteriori error estimates that are based on information obtained during the solution process itself (KELLY, GAGO & ZIENKIEWICZ [1983]; BABUSKA & GUI [1985]; SZABO [1984]; and SPECHT [1984]). For structural mechanics problems, all these error estimates were developed for compatible displacement models.

Among the error estimators developed to date are the following two:

1. Local energy norm error. This is the square root of the strain energy of the error. This is a local-global measure in the sense that it measures a global response characteristic, locally (within an individual element). In nonlinear problems, the measure can be used by linearization around a nonlinear solution and evaluating the energy norm of the linearized problem.

2. Interior and boundary residuals. These represent the equilibrium defects in the interior on the portion of the boundary where tractions are prescribed as well as the jumps in the tractions at interelement boundaries. For uniform grids with linear, bilinear and trilinear shape functions the contributions of the jumps dominate the residual and, therefore, the residual can be approximated by the traction jumps. A simple approximate method of evaluating these residuals for elements with hierarchic shape functions was given in KELLY, GAGO & ZIENKIEWICZ [1983].

The error estimators, in addition to providing information about the quality of solutions, form the basis for adaptive improvement of the finite-element solution. This can be accomplished by enriching or improving the approximation using one of the following approaches (or possibly, their combinations).

1. Refining the mesh
2. Moving the nodes (node relocation)
3. Increasing the local order of the approximation
4. Using the iterated defect correction method

The third approach has the advantages over the first two of being easy to implement and of providing a simple formula for the error estimator. The fourth approach is based on using the numerical solution obtained to construct a pseudo or neighboring problem whose exact solution is known (e.g., polynomial or spline interpolation of the discrete numerical solution). The pseudo problem is then solved using the same finite-element model as that used for the original problem. The error in the pseudo problem is assumed to be a close approximation of the error in the original problem and is used as a correction to that solution. The technique has been successfully applied to the numerical solution of stiff systems of ordinary differential equations and appears to have high potential for application to finite-element boundary value-problems (ZADUNAISKY [1976]; FRANK, HERTLING & MONNET [1983]; and BOHMER & STETTER [1984]).

6. POSTPROCESSING AND STRESS CALCULATION

In displacement finite-element models, the strain energy of the structure is the highest quality information that can be extracted from the finite-element solution. The accuracy and rate of convergence of stresses depend on how (and where) they are computed. Several approaches have been suggested for improving the accuracy of stress calculations (see, for example, HINTON & CAMPBELL [1974]; CAREY [1982]; ZIENKIEWICZ, XI-KUI & NAKAZAWA [1985]; and BABUSKA & MILLER [1984]). Among these are:

1. Evaluating the stresses at numerical quadrature points and determining their values at the nodes by extrapolation
2. Computing the stresses using the discarded structural equations (corresponding to prescribed displacement boundary conditions)
3. Averaging or smoothing based on projection techniques
4. Using influence function methods

The first approach is by far the most commonly used. Superconvergence (increased accuracy and improved rates of convergence) has been observed for stresses evaluated at quadrature points. Note that the stresses cannot have a faster convergence than the strain energy. The second approach is particularly useful for evaluating the stresses at the boundaries. The third and fourth approaches improve the accuracy of stress predictions through the filtering of spurious oscillations. A systematic assessment of the latter two approaches is needed.

7. FUTURE DIRECTIONS FOR RESEARCH

Global-local analysis strategies have high potential for the reliable and efficient prediction of the nonlinear response of complex structures subjected to different loadings. To realize this potential the global-local strategy must include the following seven key elements:

1. Rational selection of a hierarchy of mathematical models for different parts of the structure and a strategy for the adaptive refinement of these models
2. Use of global (or macro) elements for discretization whenever appropriate with interface conditions satisfied via exterior penalty method
3. Use of reliable failure criteria and discrete elements that account for the progressive failure mechanisms
4. Application of operator splitting in conjunction with reduction method for generating the response of the complex structure by using large perturbations from the response of a simpler structure
5. Postprocessing to increase the accuracy of stress calculations
6. Quality control of numerical solutions
7. Exploiting the computational power of new multiprocessor machines through parallelization of the problem formulation, computational strategy as well as the numerical algorithms.

Each of the aforementioned key elements requires major development to reach the level of maturity needed for routine inclusion in the global-local strategy. To this end, there are pacing items that must be addressed by the research community. Among the items that pace the progress of global-local methodologies are the development of:

1. Criteria and control parameters for selecting the mathematical model, as well as adaptive strategies for refining the model whenever needed. Also, strategies for blending regions of different structural behavior (e.g., boundary layer, two/three dimensional models of the structure).
2. Reliable failure criteria and shell elements that account for the composite delamination mechanisms.
3. Simple and accurate techniques for stress calculations which provide the same accuracy as that of the strain energy.
4. Error estimation and adaptive improvement strategies. This is an area which requires more attention by researchers. In particular, error estimators that satisfy the following four criteria need to be developed for nonlinear analysis:
 - a) provide reliable local assessment of the error with extrapolation to global estimation
 - b) computationally inexpensive to evaluate
 - c) applicable to a wide class of discrete elements
 - d) easy to use in conjunction with adaptive improvement
5. Parallel computational strategies for multiprocessor computers.

These strategies include the use of: a) primitive variables (e.g., three-field mixed formulation); b) domain decomposition (with minimization of interfaces); and c) operator splitting to uncouple the algebraic equations.

In addition, the intense research effort currently under way on parallel numerical algorithms (see, for example, NOOR [1983], and PADDON [1984]) should be brought to bear on global-local methodologies. Due to the wide variety of new parallel computers, the idea of developing macro algorithms which are efficient on different parallel machines should be investigated. The numerical tasks in these algorithms are performed by different programs which are optimized for each of the individual machines.

CONCLUDING REMARKS

A review and an assessment were made of global-local strategies for the nonlinear analysis of structures. To provide a focus for the discussion two benchmark problems of postbuckling of laminated composite cylindrical panels were selected. The major characteristics of these problems were identified.

A number of global-local analysis strategies were reviewed, their potential for solving the benchmark problems discussed and their shortcomings delineated. Also, error estimation and postprocessing techniques were reviewed.

The items that pace the progress of global-local methodologies are identified and are, therefore, recommended as future directions for research. These include the coupling of different global-local methodologies; postprocessing and stress calculation methods; quality control and adaptive improvement of numerical algorithms; and effective computational strategies for new computing systems.

REFERENCES

- BABUSKA, I. & MILLER, A. (1984), "The Postprocessing Approach in the Finite Element Method - Part 1: Calculation of Displacements, Stresses, and Other Higher Derivatives of the Displacements," *International Journal for Numerical Methods in Engineering*, Vol. 20, pp. 1085-1109.
- BABUSKA, I. & GUI, W. (1985), "Basic Principles of Feedback and Adaptive Approaches in the Finite Element Method," Technical Note BN-1042, Institute for Physical Science and Technology, University of Maryland.
- BELYTSCHKO, T. & HUGHES, T. J. R. (eds.) (1983), Computational Methods for Transient Analysis, North Holland Publishing Company.
- BOHMER, K. & STETTER, H. J. (eds.) (1984), Defect Correction Methods - Theory and Applications, Springer-Verlag.
- CAREY, G. F. (1982), "Derivative Calculations From Finite Element Solutions," *Computer Methods in Applied Mechanics and Engineering*, Vol. 35, pp. 1-14.
- DELVES, L. M. & HALL, C. A. (1979), "An Implicit Matching Principle for Global Element Calculations," *Journal of the Institute of Mathematics and Applications*, Vol. 23, pp. 223-234.

- DELVES, L. M. & PHILLIPS, C. (1980), "A Fast Implementation of the Global Element Method," *Journal of the Institute of Mathematics and Its Applications*, Vol. 25, pp. 177-197.
- DONG, S. B. (1983), "Global-Local Finite Element Methods," in State-of-the-Art Surveys on Finite Element Technology, ed. by A. K. Noor and W. D. Pilkey, American Society of Mechanical Engineers, pp. 451-474.
- FELIPPA, C. A. (1978), "Iterative Procedures for Improving Penalty Function Solutions of Algebraic Systems," *International Journal for Numerical Methods in Engineering*, Vol. 12, pp. 821-836.
- FRANK, R., HERTLING, J. & MONNET, J. P. (1983), "The Application of Iterated Defect Correction to Variational Methods for Elliptic Boundary Value Problems," *Computing*, Vol. 30, pp. 121-135.
- GOLDENVEIZER, A. L. (1976), Theory of Elastic Thin Shells, Nauka.
- HACKBUSCH, W. & TROTTEBERG, W. (eds.) (1982), Multigrid Methods, Lecture Notes in Mathematics No. 960. Proceedings of the conference held at Köln-Portz, Germany, Nov. 23-27, 1981, Springer-Verlag, Berlin.
- HINTON, E. & CAMPBELL, J. (1974), "Local and Global Smoothing of Discontinuous Finite Element Functions Using a Least Square Method," *International Journal for Numerical Methods in Engineering*, Vol. 8, pp. 461-480.
- HIRAI, I., UCHIYAMA, Y., MIZUTA, Y. & PILKEY, W. D. (1985), "An Exact Zooming Method," *Finite Elements in Analysis and Design*, Vol. 1, pp. 61-69.
- HUGHES, T. J. R., RAEFSKY, A., MULLER, A., WINGET, J. & LEVIT, I. (1984), "A Progress Report on EBE Solution Procedures in Solid Mechanics," Proceedings of the Second International Conference on Numerical Methods for Nonlinear Problems, Barcelona, Spain, April 9-13, 1984.
- IYENGAR, K. T. S. R., CHANDRASHEKHARAN, K. & SEBASTIAN, V. K. (1974), "On the Analysis of Thick Rectangular Plates," *Ingenieur-Archiv*, Vol. 43, pp. 317-330.
- KAMIYA, N. & SAWAKI, Y. (1982), "An Integral Equation Approach to Finite Deflection of Elastic Plates," *International Journal of Nonlinear Mechanics*, Vol. 17, No. 3, pp. 187-194.
- KAMIYA, N., SAWAKI, Y. & NAKAMURA, Y. (1984), "Postbuckling Analysis by the Boundary Element Method," *Engineering Analysis*, Vol. 1, No. 1, pp. 40-44.
- KARDESTUNCER, H. (ed.) (1985), Unification of Finite Element Software Systems, North-Holland Publishing Company.
- KELLY, D. W., GAGO, J. P. de S. R. & ZIENKIEWICZ, O. C. (1983), "A Posteriori Error Analysis and Adaptive Processes in the Finite Element Method: Part I - Error Analysis; Part II - Adaptive Mesh Refinement," *International Journal for Numerical Methods in Engineering*, Vol. 19, pp. 1593-1619 and 1621-1656.

- KNIGHT, N. F., GREENE, W. H. & STROUD, W. J. (1985), "Nonlinear Response of a Blade-Stiffened Graphite-Epoxy Panel with a Discontinuous Stiffener: Work in Progress," Proceedings of the Workshop on Computational Methods of Structural Mechanics and Dynamics, NASA Langley Research Center, Hampton, VA, June 19-21, 1985.
- KNIGHT, N. F. & STARNES, J. H. (1984), "Postbuckling Behavior of Axially Compressed Graphite-Epoxy Cylindrical Panels with Circular Holes," in Collapse Analysis of Structures, ed. by L. H. Sobel and K. Thomas, PVP-Vol. 84, American Society of Mechanical Engineers, pp. 153-167.
- LIU, W. K., BELYTSCHKO, T. & PARK, K. C. (1984), Innovative Methods for Nonlinear Problems, Pineridge Press, Swansea, United Kingdom.
- MIKLOSKO, J. & KOTOV, V. E. (eds.) (1984), Algorithms, Software and Hardware of Parallel Computers, Springer-Verlag.
- MOTE, C. D. (1971), "Global-Local Finite Element," International Journal for Numerical Methods in Engineering, Vol. 3, pp. 565-574.
- NOOR, A. K. (1982), "On Making Large Nonlinear Problems Small," Computer Methods in Applied Mechanics and Engineering, Vol. 34, pp. 955-985.
- NOOR, A. K. (ed.) (1983), Impact of New Computing Systems on Computational Mechanics, The American Society of Mechanical Engineers.
- NOOR, A. K. (1985), "Reduction Method for the Nonlinear Analysis of Symmetric Anisotropic Panels," Proceedings of the Europe-U.S. Symposium on Finite Element Methods for Nonlinear Problems, Norwegian Institute of Technology, Trondheim, Norway, Aug. 12-15, 1985.
- NOOR, A. K. & ANDERSEN, C. M. (1982), "Mixed Models and Reduced/Selective Integration Displacement Models for Nonlinear Shell Analysis," International Journal for Numerical Methods in Engineering, Vol. 18, pp. 1429-1454.
- NOOR, A. K. & PETERS, J. M. (1983), "Recent Advances in Reduction Methods for Instability Analysis of Structures," Computers and Structures, Vol. 16, pp. 67-80.
- NOOR, A. K. & PETERS, J. M. (1985), "Model-Size Reduction Technique for the Analysis of Symmetric Anisotropic Structures," Engineering Computations, Vol. 2, No. 4, pp. 285-292.
- NOOR, A. K. & PILKEY, W. D. (eds.) (1983), State-of-the-Art Surveys on Finite Element Technology, The American Society of Mechanical Engineers.
- NOOR, A. K., ANDERSEN, C. M. & TANNER, J. A. (1985), "Exploiting Symmetries in the Modeling and Analysis of Tires," Proceedings of the Workshop on Computational Methods of Structural Mechanics and Dynamics, NASA Langley Research Center, Hampton, Virginia, June 19-21, 1985.

- NOOR, A. K., MATHERS, M. D. & ANDERSON, M. S. (1977), "Exploiting Symmetries for Efficient Postbuckling Analysis of Composite Plates," AIAA Journal, Vol. 15, No. 1, pp. 24-32.
- NOOR, A. K., STORASSLI, O. O. & FULTON, R. E. (1984), "Impact of New Computing Systems on Computational Mechanics and Flight Vehicle Structures Technology," AGARD Report No. 706.
- PADDON, D. J. (ed.) (1984), Supercomputers and Parallel Computation, Clarendon Press, Oxford.
- PATTIBIRAMAN, J., RAMAMURTI, V. & REDDY, D. V. (1974), "Statics and Dynamics of Elastic Shells with Cutouts - A Review," Journal of Ship Research, Vol. 18, No. 2, pp. 113-126.
- SCHENDEL, U. (1984), Introduction to Numerical Methods for Parallel Computers, Ellis Horwood, Ltd.
- SPECHT, B. (1984), "A General Construction of Local Error Estimates for Conforming Finite Elements," Computers and Structures, Vol. 19, pp. 815-822.
- STANLEY, G. M. (Aug. 1985), "Continuum-Based Shell Elements," Ph.D. Dissertation, Stanford University.
- SZABO, B. A. (1984), "Estimation and Control of Error Based on p-Convergence," Proceedings of the International Conference on Accuracy Estimates and Adaptive Refinements in Finite Element Computations, Lisbon, Portugal, June 19-22, 1984.
- UNDERWOOD, P. G. (1983), "Dynamic Relaxation - A Review," in Computational Methods for Transient Dynamic Analysis, ed. by T. Belytschko and T. J. R. Hughes, North Holland Publishing Company.
- UTKU, S. & MELOSH, R. J. (1984), "Solution Errors in Finite Element Analysis," Computers and Structures, Vol. 18, pp. 379-393.
- VLASOV, V. Z. & LEONTEV, U. N. (1966), "Beams, Plates and Shells on Elastic Foundations," NASA TT-F-357.
- WILKINS, D. J. (1983), "A Preliminary Damage Tolerance Methodology for Composite Structures," in Failure Analysis and Mechanisms of Failure of Fibrous Composite Structures, NASA CP-2278, pp. 67-93.
- ZADUNAISKY, P. E. (1976), "On the Estimation of Errors Propagated in the Numerical Integration of Ordinary Differential Equations," Numer. Math., Vol. 27, pp. 21-39.
- ZIENKIEWICZ, O. C., XI-KUI, L. & NAKAZAWA, S. (1985), "Iterative Solution of Mixed Problems and the Stress Recovery Procedures," Communications in Applied Numerical Methods, Vol. 1, pp. 3-9.

TABLE 1. CHARACTERISTICS OF THE FINITE-ELEMENT MODELS USED FOR THE BENCHMARK PROBLEMS

	Unstiffened Panel	Blade-Stiffened Panel (Knight, Greene & Stroud [1985])	
		Mesh 1	Mesh 2
Type of Element	Mixed, 9-Noded	← Hybrid 4-Noded →	
Number of Elements	66	188 (376)	544 (1088)
Number of Displacement Degrees of Freedom	1338	1266 (2432)	3504 (7008)

TABLE 2. PARTIAL LIST OF THE COMMONLY USED TECHNIQUES FOR GLOBAL AND LOCAL ANALYSES

GLOBAL (APPROXIMATE) ANALYSIS	LOCAL (DETAILED) ANALYSIS
<ul style="list-style-type: none"> o Global variational methods o Discrete element methods <ul style="list-style-type: none"> o Conventional finite elements o Global element method o Boundary element method o Analytic solutions 	<ul style="list-style-type: none"> o Discrete element methods <ul style="list-style-type: none"> o Conventional finite elements o Special elements o Superelements o Boundary element method o Analytic solutions

16-PLY UNSTIFFENED PANEL

BLADE STIFFENED PANEL

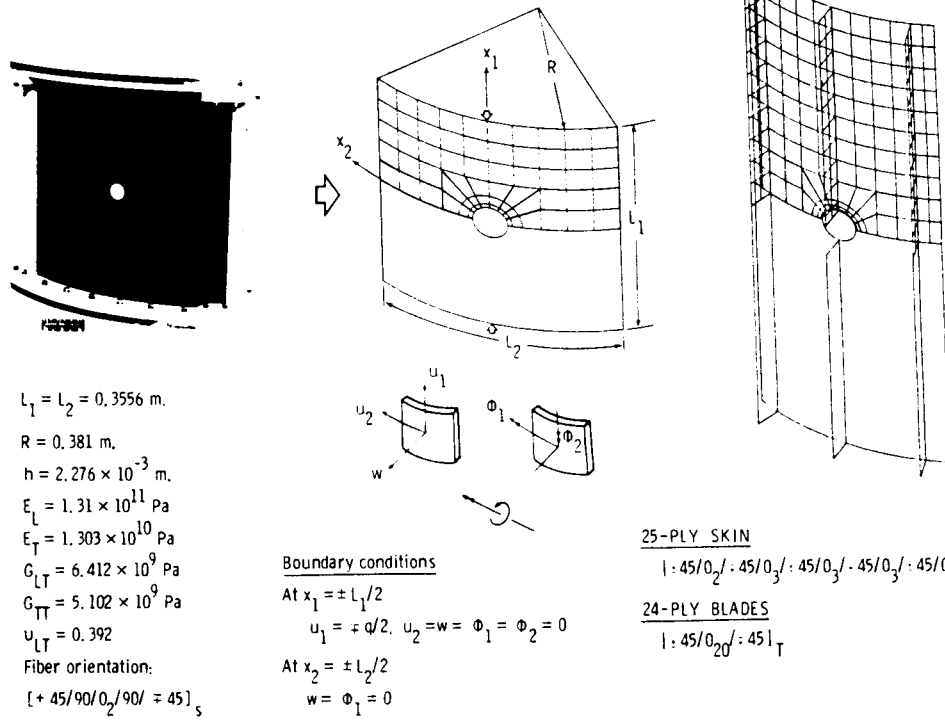


Figure 1. Benchmark problems considered in present study.

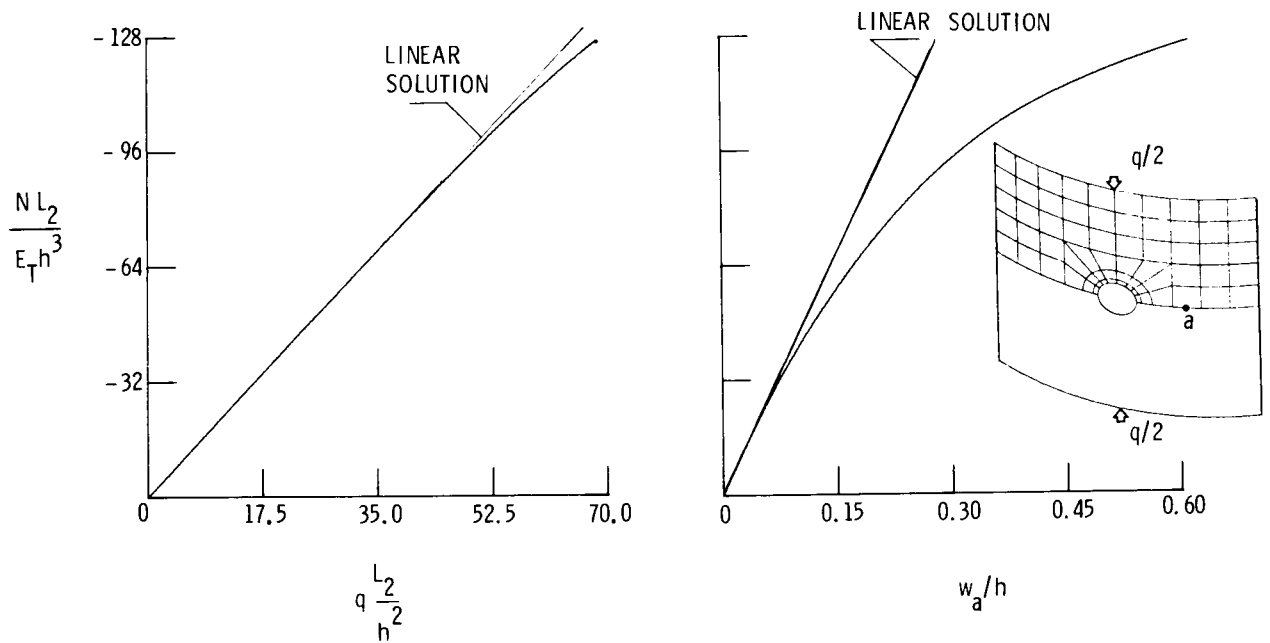


Figure 2. Nonlinear response of unstiffened panel subjected to end shortening (see fig. 1).

ORIGINAL PAGE IS
OF POOR QUALITY

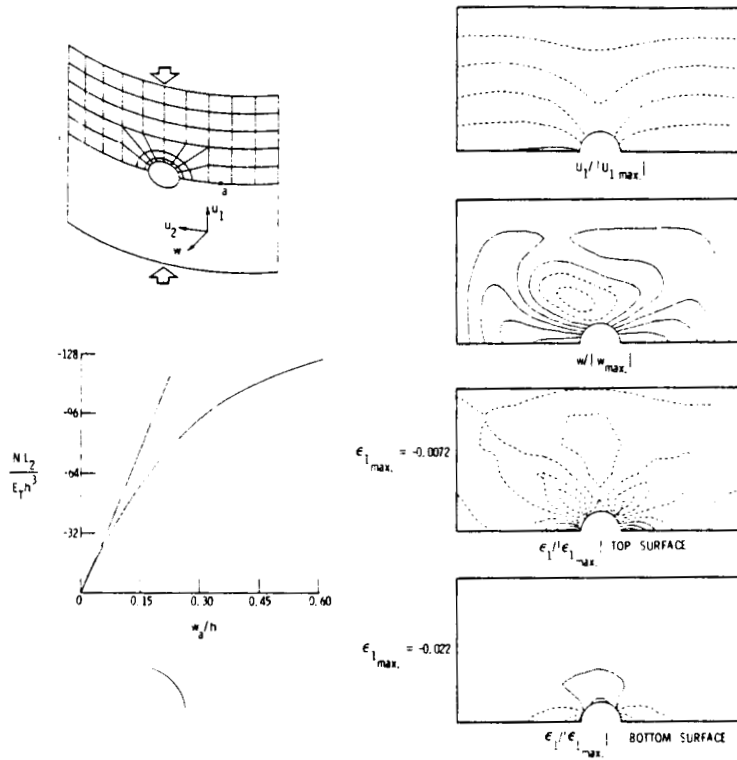


Figure 3. Normalized contour pilots for displacements and strains in unstiffened panel at $\frac{NL_2}{E_T h^3} = 126.6$ (see fig. 1).

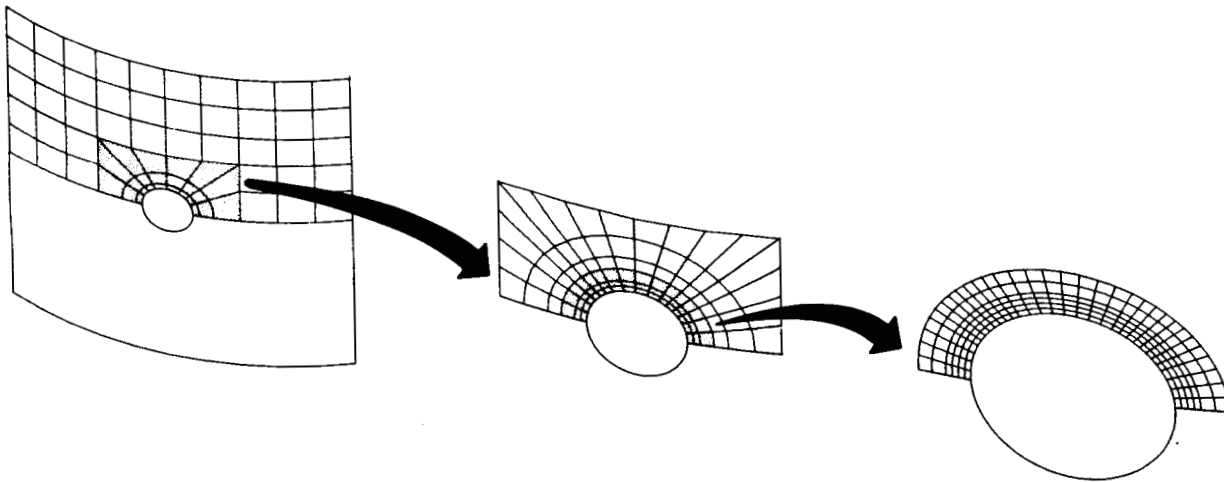
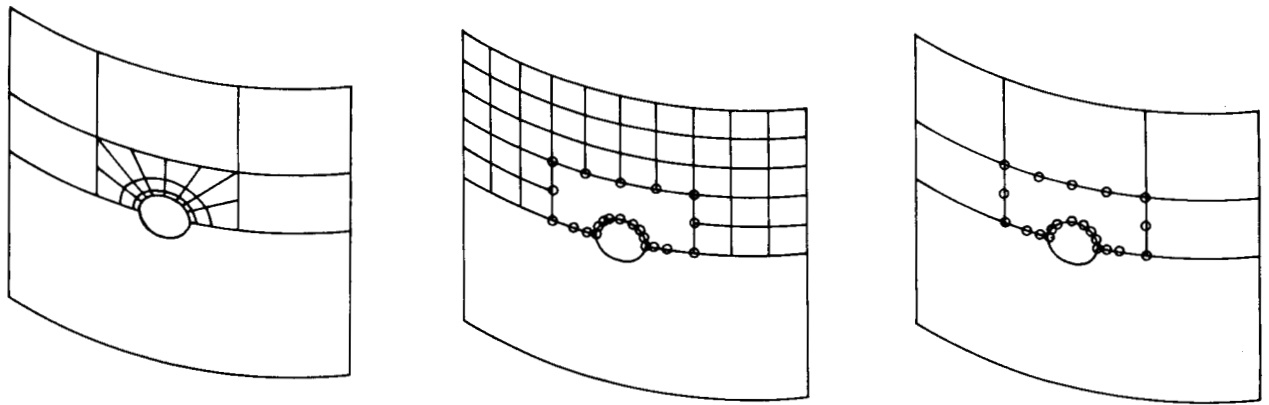


Figure 4. Application of zooming technique to unstiffened panel.



GLOBAL ANALYSIS	GLOBAL ELEMENTS	CONVENTIONAL FINITE ELEMENTS	GLOBAL ELEMENTS
LOCAL ANALYSIS	CONVENTIONAL (OR SPECIAL) FINITE ELEMENTS	BOUNDARY ELEMENTS	BOUNDARY ELEMENTS

Figure 5. Simultaneous application of two discrete element methods to the analysis of unstiffened panel.

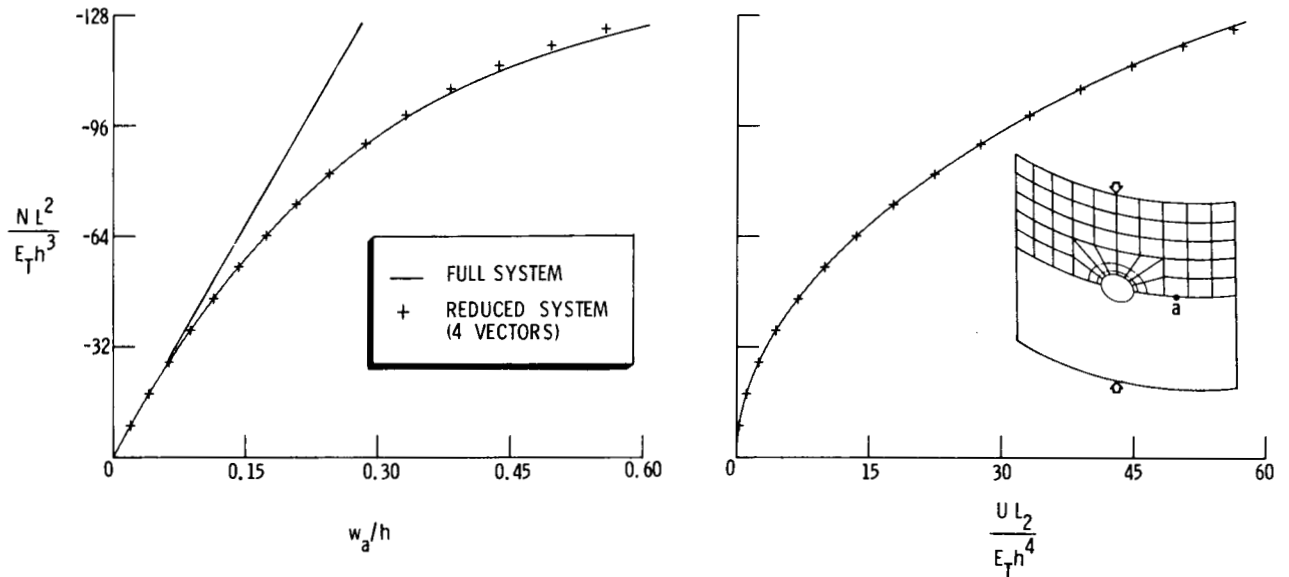


Figure 6. Accuracy of normal displacement w and total strain energy U obtained by reduction method. Unstiffened panel subjected to axial end shortening (see fig. 1).

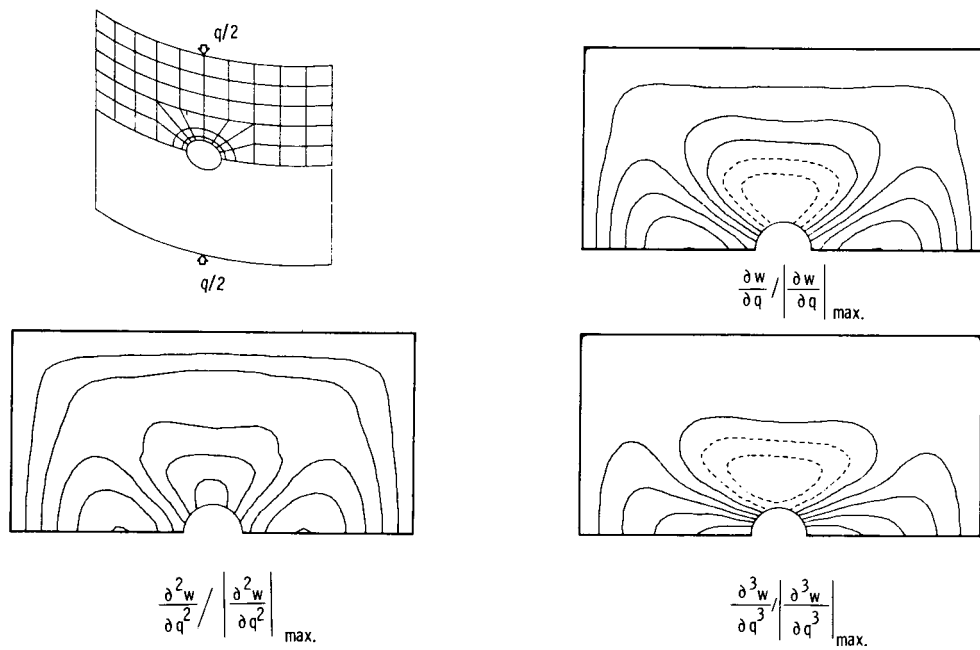


Figure 7. Normalized contour plots for the global approximation vectors - unstiffened panel subjected to axial end shortening (see fig. 1).

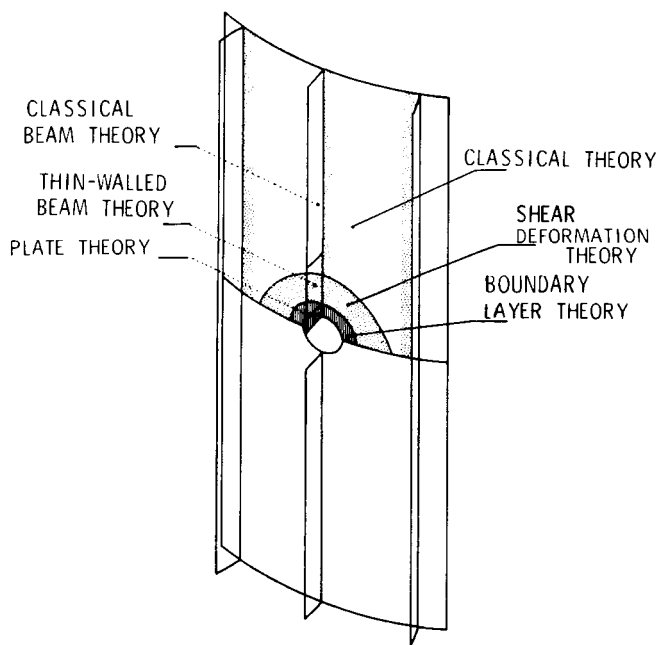


Figure 8. Heuristic approach for selecting a hierarchy of mathematical models for blade stiffened panel.

APPLICATION OF THE P-VERSION OF THE FINITE-ELEMENT METHOD
TO GLOBAL-LOCAL PROBLEMS

Barna A. Szabó

Washington University

St. Louis, MO

1. INTRODUCTION

The following is a brief survey of some recent developments in finite-element analysis technology which bear upon the three main research areas under consideration in this workshop: (1) analysis methods; (2) software testing and quality assurance; and (3) parallel processing.

The variational principle incorporated in a finite-element computer program, together with a particular set of input data, determines the exact solution corresponding to that input data. Most finite-element analysis computer programs are based on the principle of virtual work. In the following we consider only programs based on the principle of virtual work and denote the exact displacement vector field corresponding to some specific set of input data by \bar{u}_{EX} . The exact solution \bar{u}_{EX} is independent of the design of the mesh or the choice of elements. Except for very simple problems, or specially constructed test problems, \bar{u}_{EX} is not known.

We perform a finite-element analysis (or any other numerical analysis) because we wish to make conclusions concerning the response of a physical system to certain imposed conditions, as if \bar{u}_{EX} were known. We know the finite-element solution only which we denote by \bar{u}_{FE} . The solution \bar{u}_{FE} depends not only on the variational principle and the input data but also on the finite-element mesh and the choice of elements. We will assume that the finite-elements are exactly and minimally conforming and therefore the elements are completely characterized by their polynomial degree. We therefore control \bar{u}_{FE} by mesh design and the choice of the polynomial degree of elements.

We wish to compute \bar{u}_{FE} so that \bar{u}_{FE} is close to \bar{u}_{EX} in some sense. For example, if we are interested in determining a stress intensity factor then we wish to have the stress intensity factor computed from \bar{u}_{FE} to be close to the stress intensity factor computed from \bar{u}_{EX} within some prespecified level of tolerance τ . In general, we wish to determine functionals $\Psi_i(\bar{u}_{FE})$ ($i = 1, 2, \dots, n$) so that:

$$\left| \frac{\Psi_i(\bar{u}_{EX}) - \Psi_i(\bar{u}_{FE})}{\Psi_i(\bar{u}_{EX})} \right| \leq \tau_i \quad (i = 1, 2, \dots, n) \quad (1)$$

The question naturally arises: how can we tell whether $\Psi_i(\bar{u}_{FE})$ is close to $\Psi_i(\bar{u}_{EX})$ if we do not know \bar{u}_{EX} ? The answer is: by performing *extensions*. Both the estimation and control of error are based on extensions.

Extensions are systematic increases in the number of degrees of freedom either by mesh refinement, increase in the polynomial degree of elements or a combination of both. If the extension is by mesh refinement then the process is called *h-extension**. If the extension is by increase in the polynomial degree of elements then the process is called *p-extension*** . If the extension is by a combination of proper mesh refinement and concurrent increase in the polynomial degree of elements then it is called *h-p extension*. Having performed an extension, we may draw conclusions concerning the overall quality of the approximate solution and the quality of any functional computed from \bar{u}_{FE} .

2. OVERALL QUALITY

The overall quality of approximation can be judged in terms of the estimated error in energy norm and errors in equilibrium. Estimation of error in energy norm is outlined in some detail and an example is presented. Procedures for assessment of the quality of approximation in terms of errors in equilibrium are briefly discussed.

2.1. Estimation of error in energy norm.

We know that the strain energy of the error $U(\bar{u}_{EX} - \bar{u}_{FE})$ must decrease monotonically as we systematically refine the mesh or increase the polynomial degree of elements and a well developed, elaborate theoretical basis exists for the estimation of error in energy norm for the h-, p- and h-p extension processes. (See, for example, [1,2,3,4,5].) The error in energy norm is defined as:

$$\|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)} \equiv \sqrt{U(\bar{u}_{EX} - \bar{u}_{FE})} \quad (2)$$

where Ω represents the solution domain and U represents the strain energy. $\|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)}$ is closely related to the root-mean-square of error in stresses [6].

In the case of h- and p-extensions the estimate is of the form:

$$\|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)} \leq \frac{k}{N^\beta} \quad (3)$$

where k and β are positive constants, N is the number of degrees of freedom. In the case of h-p extensions the estimate is of the form:

$$\|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)} \leq \frac{k}{\exp(\gamma N^\theta)} \quad (4)$$

where k , γ and θ are positive constants. These estimators are 'sharp' for large N values hence the 'less than or equal' (\leq) can be replaced by 'approximately equal' (\approx) in (3), (4) when N is large. Therefore from (3) for large N values we have:

$$\log \|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)} \approx \log k - \beta \log N \quad (5)$$

If we plot $\log \|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)}$ versus $\log N$ we see a downward sloping straight line. The absolute value of the slope is β , called the *asymptotic rate of convergence*. When β is large then the error decreases rapidly as N is increased. When β is small then the error decreases slowly. Of course, the error also depends on k which

* h represents the size of elements. *h-Extension* involves letting $h_{max} \rightarrow 0$.

** p represents the polynomial degree of elements. *p-Extension* involves letting $p_{min} \rightarrow \infty$.

is generally not known a priori, but can be estimated from data obtained from properly performed extensions. This will be discussed later. When the estimate is of the form (3) the rate of convergence is said to be *algebraic*.

When the estimate is of the form (4) and we plot $\log \|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)}$ versus $\log N$ then for large N values we see a downward curving line [2,3,4,5,7]. In this case the rate of convergence is *exponential*:

$$\log \|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)} \approx \log k - \gamma (\log e) N^\theta \quad (6)$$

where e is the base of the natural logarithm. If we plot $\log \|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)}$ versus N^θ (not $\log N$ as before) then we see a downward sloping straight line. It is known that under conditions which are generally satisfied in practice $\theta \geq 1/3$ [5].

All error estimation techniques are based on extension. Because in general the exact solution \bar{u}_{EX} is not known, the only information available to us is how the finite-element solution \bar{u}_{FE} behaves when the number of degrees of freedom is increased either through mesh refinement or increase in the polynomial degree of elements. Such information, together with an estimate or hypothesis concerning the magnitude of the error, or its rate of change with respect to N , is essential to all error estimation. Of course, the estimate or hypothesis must be asymptotically correct: as $N \rightarrow \infty$ the estimated error must approach zero at the same rate as the true error does. Therefore the quality of error estimators should increase with N .

P-extension makes it convenient and inexpensive to obtain information concerning the rate of change of $U(\bar{u}_{FE})$ with respect to N . In the p-version *hierarchical* basis functions are used. Therefore the stiffness matrices and load vectors corresponding to polynomial degree p are embedded in the stiffness matrices and load vectors of polynomial degree $p+1$. Once a solution is available for polynomial degree p_{max} , all solutions corresponding to $p = 1, 2, \dots, p_{max} - 1$ can be readily and inexpensively obtained. Specifically, we write:

$$\|\bar{u}_{EX} - \bar{u}_{FE}\|_{E(\Omega)}^2 = U(\bar{u}_{EX} - \bar{u}_{FE}) = |U(\bar{u}_{EX}) - U(\bar{u}_{FE})| \approx \frac{k^2}{N^{2\beta}} \quad (7)$$

Let us assume for the moment that $U(\bar{u}_{EX}) > U(\bar{u}_{FE})$. In that case:

$$U(\bar{u}_{EX}) - U(\bar{u}_{FE}) \approx \frac{k^2}{N^{2\beta}} \quad (8)$$

We have three unknowns: $U(\bar{u}_{EX})$, k and β . If we have three values of $U(\bar{u}_{FE})$ and N corresponding to three different values of p , then we have three equations for computing the unknowns. Let us denote these three values by U_p, U_{p-1}, U_{p-2} and N_p, N_{p-1}, N_{p-2} and $U(\bar{u}_{EX})$ by U . Then from (8) we have:

$$\frac{\log \frac{U - U_p}{U - U_{p-1}}}{\log \frac{U - U_{p-1}}{U - U_{p-2}}} \approx \frac{\log \frac{N_{p-1}}{N_p}}{\log \frac{N_{p-2}}{N_{p-1}}} \quad (9)$$

Denoting the right hand side of (9) by Q , we have:

$$\frac{U - U_p}{U - U_{p-1}} \approx \left(\frac{U - U_{p-1}}{U - U_{p-2}} \right)^Q \quad (10)$$

To obtain an estimate of the exact strain energy U , we need to solve (10). The solution is expected in the neighborhood of U_p . Because convergence of the strain

energy is monotonic, we know that $U \geq U_p$ when $U_p > U_{p-1}$. Conversely, $U \leq U_p$ when $U_p < U_{p-1}$. Eq.(10) would not be different if $U(\bar{u}_{EX}) < U(\bar{u}_{FE})$; therefore, the restriction that $U(\bar{u}_{EX}) > U(\bar{u}_{FE})$ is not essential. Computational experience has shown this estimate to be reliable and generally accurate, with the accuracy of the estimate increasing with the accuracy of U_p .

2.2. Example.

The following test problem is representative of plate and shell intersections and reentrant corner problems in general. An L-shaped plane elastic body of thickness t is loaded by tractions. The tractions are computed from a stress field which satisfies the equilibrium and compatibility equations and the stress free conditions along the reentrant edges. Specifically, the stress field corresponds to the first (symmetric or 'Mode 1') term of the asymptotic expansion of \bar{u}_{EX} about the reentrant corner. (See, for example, [8].) Therefore the exact solution is known. Specifically, the components of \bar{u}_{EX} in the coordinate system shown in Fig. 1 are:

$$u_x = \frac{A}{2G} r^\lambda [(\kappa - Q(\lambda + 1)) \cos \lambda \theta - \lambda \cos(\lambda - 2) \theta] \quad (11a)$$

$$u_y = \frac{A}{2G} r^\lambda [(\kappa + Q(\lambda + 1)) \sin \lambda \theta + \lambda \sin(\lambda - 2) \theta] \quad (11b)$$

where A is a generalized stress intensity factor; $\lambda = 0.544483737$; $Q = 0.543075579$; G is the modulus of rigidity and κ depends on Poisson's ratio ν only. For plane strain: $\kappa = 3 - 4\nu$. We assume plane strain conditions and $\nu = 0.3$, therefore in this case $\kappa = 1.8$.

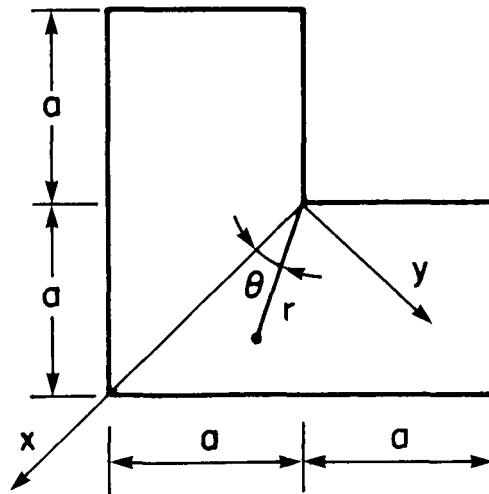


Fig. 1. L-shaped plane elastic body.

The stress tensor components are:

$$\sigma_x = A \lambda r^{\lambda-1} [(2 - Q(\lambda + 1)) \cos(\lambda - 1) \theta - (\lambda - 1) \cos(\lambda - 3) \theta] \quad (12a)$$

$$\sigma_y = A \lambda r^{\lambda-1} [(2 + Q(\lambda + 1)) \cos(\lambda - 1) \theta + (\lambda - 1) \cos(\lambda - 3) \theta] \quad (12b)$$

$$\tau_{xy} = A \lambda r^{\lambda-1} [(\lambda - 1) \sin(\lambda - 3) \theta + Q(\lambda + 1) \sin(\lambda - 1) \theta]. \quad (12c)$$

Because we know the exact displacement and stress fields we can compute the strain energy of the exact solution:

$$U(\tilde{u}_{EX}) = 4.15454423 \frac{A^2 a^{2\lambda} t}{E} \quad (13)$$

where E is the modulus of elasticity. The relative error in energy norm is defined as follows:

$$(e_r)_E = \sqrt{\frac{|U(\tilde{u}_{EX}) - U(\tilde{u}_{FE})|}{U(\tilde{u}_{EX})}} \quad (14)$$

Using the mesh shown in Fig. 2 finite-element solutions were obtained for $p=1$ to 8. The computations were performed by a new computer program, called *PROBE* [9]. The number of degrees of freedom, the computed strain energy, the estimated and true relative errors in energy norm, computed from eq. (14), are shown in Table 1.

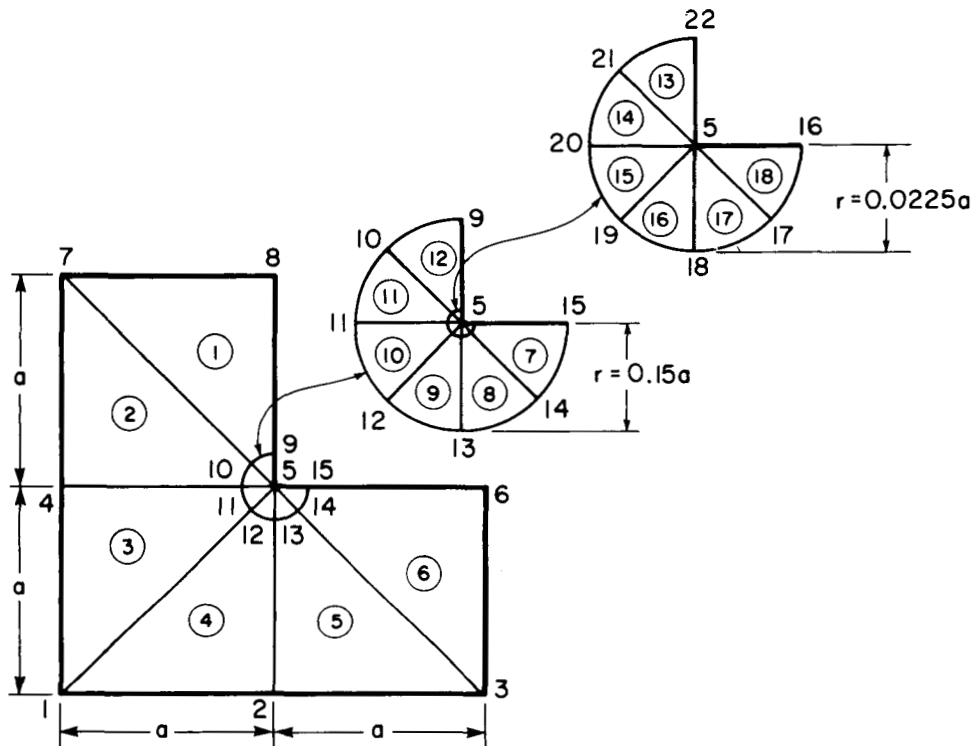


Fig. 2. Mesh design.

The results presented in Table 1 are typical of the quality of the error estimate we can obtain by means of the procedure described in Section 2.1. When the mesh is strongly graded toward the point of singularity then the convergence path (the $\log(e_r)_E$ versus $\log N$ curve) looks like an inverted S [3,4,5,10]. For low N values the rate of convergence is nearly exponential and the downward slope increases with N . In this segment the estimated error is conservative. Near the inflection

Table 1. Estimated and true relative error in energy norm.

p	N	$\frac{U(\bar{u}_{FE})E}{A^2 a^{2\lambda} t}$	2β	$Est.'d$ $(e_r)_E$	$True$ $(e_r)_E$
1	41	3.8860880	—	—	25.42
2	119	4.1248326	—	—	8.46
3	209	4.1481150	1.93	5.34	3.93
4	335	4.1526504	2.76	2.02	2.14
5	497	4.1536354	3.05	1.01	1.48
6	695	4.1539746	2.45	0.80	1.17
7	929	4.1541390	1.83	0.75	0.99
8	1199	4.1542378	1.39	0.75	0.86
∞	∞	4.1545442	1.09	—	0

point, (i.e. where the curvature of the convergence path changes from negative to positive) the estimate is the least accurate and not conservative, nevertheless as we see in this example, it remains close. The estimate then becomes progressively more accurate as the asymptotic range of the p-extension is entered. In this case the correct asymptotic rate of convergence is $\beta = \lambda = 0.5445$. At $p = 8$ the computed value of β is approximately 0.7 with β decreasing.

2.3. Equilibrium tests.

Smallness of error in energy norm is a necessary but not sufficient condition for ensuring that the overall quality of the finite-element solution is good. It is possible to produce examples where the estimated error in energy norm is small (under 1 percent) yet the error in overall equilibrium is large (well over 10 percent).

Although we do not know \bar{u}_{EX} we know that \bar{u}_{EX} satisfies the equations of equilibrium and the law of action and reaction. We can, therefore, assess the quality of the finite-element solution by examining to what degree \bar{u}_{FE} satisfies equilibrium and the law of action and reaction. Specifically, we can perform: (1) overall equilibrium tests; (2) element by element equilibrium tests and (3) action-reaction tests.

In the *overall equilibrium test* we 'cut' the structure from its supports and integrate the tractions, computed from the \bar{u}_{FE} , to obtain the reactions. In this way a free body diagram is produced. The error in equilibrium must be small in relation to the magnitude of applied forces. For example, we can define:

$$F = \sqrt{\sum_{i=1}^N r_i^2} \quad (15)$$

where r_i are the (global) load vector components. The term F is a suitable measure of the magnitude of the applied load.

In the *element-by-element equilibrium test* individual elements (or any group of elements) are separated from the model and tested for equilibrium. Specifically, we denote the element domain by Ω_e and its boundary by $\partial\Omega_e$. We compute:

$$q_i^{(e)} = \int_{\Omega_e} (\sigma_{ij,j} + X_i) t dx_1 dx_2 + \int_{\partial\Omega_e} \sigma_{ij} n_j t ds \quad (i, j = 1, 2) \quad (16)$$

where σ_{ij} are the stress components computed from the finite element solution; X_i represents the applied body force components and n_j represents the unit normal to $\partial\Omega_e$. The summation convention is used. We should have $|q_i^{(e)}| \equiv \sqrt{q_i^{(e)} q_i^{(e)}}$ as well as the absolute values of each component of both expressions on the right hand side of (16) small in relation to the magnitude of the applied loads. If $|q_i^{(e)}|$ is small but the absolute values of the integral expressions are not small then there is a local error but, according to Saint-Venant's principle, the effect of the local error will not be substantial at some distance from the element in question. If $|q_i^{(e)}|$ is large, even after p-extension was performed, then element e , and possibly its neighbors, should be subdivided. Thus the element by element equilibrium test provides information about the *quality of mesh design*. In many cases minor local refinement (for example, dividing one element into two elements) can have a highly beneficial effect on the overall quality of approximation when p-extension is used.

In the *action-reaction test* we compute the stress resultants along interelement boundaries and external element boundaries where tractions are applied. Along interelement boundaries the stress resultants computed for neighboring elements should have nearly the same absolute value and opposite sense. Along external boundaries the resultants of the applied tractions and the tractions computed from the finite element solution should be nearly the same.

Examples of equilibrium tests are presented in [11].

3. LOCAL QUALITY

Having ascertained that the overall solution quality is acceptable, we are ready to compute the quantities which are of principal interest, i.e. $\Psi_i(\bar{u}_{FE})$. Smallness of error in energy and equilibrium does not guarantee that all functionals computed from \bar{u}_{FE} are accurate. It is advisable to perform convergence tests on at least the more difficult functionals. We demonstrate the procedure by computing the direction and magnitude of the principal stresses at a point close to the reentrant corner. We selected the point $r = 0.025a$; $\theta = 30^\circ$ in the coordinate system shown in Fig. 1. The stress components computed from the exact solution are:

$$\sigma_x = 3.67198Aa^{\lambda-1} \quad \sigma_y = 7.68696Aa^{\lambda-1} \quad \tau_{xy} = 0.698375Aa^{\lambda-1} \quad (17)$$

Therefore the principal stresses σ_1 and σ_2 and the direction of the first principal stress σ_1 from the positive x-axis, denoted by θ_1 , are:

$$\sigma_1 = 7.804Aa^{\lambda-1} \quad \sigma_2 = 3.554Aa^{\lambda-1} \quad \theta_1 = 80.4^\circ \quad (18)$$

In general functionals, other than the strain energy, do not converge monotonically, nevertheless the fact that convergence has occurred should be obvious. Here σ_1 and σ_2 happen to converge monotonically but θ_1 does not. We see that the state of stress is known with sufficient accuracy for engineering purposes at $p=4$ (335 degrees of freedom, see Table 1). Extension beyond $p=4$ merely confirms that convergence has occurred to within the range of precision normally expected in engineering computations and thereby establishes reliability of the data.

This test problem demonstrates that accurate stress data can be obtained in the very close proximity of stress singularities. Other examples and additional discussion of this point are presented in [10,12].

Table 2. Principal stresses at $r = 0.025a$; $\theta = 30^\circ$.

p	$\frac{\sigma_1}{Aa^{\lambda-1}}$	$\frac{\sigma_2}{Aa^{\lambda-1}}$	θ_1 (degrees)
1	7.096	2.854	92.1
2	7.441	3.047	84.1
3	7.532	3.294	81.6
4	7.731	3.483	80.9
5	7.754	3.525	80.6
6	7.773	3.545	80.5
7	7.786	3.551	80.5
8	7.791	3.553	80.4
∞	7.804	3.554	80.4

It is possible also to compute various functionals from \bar{u}_{FE} using advanced methods of extraction [13,14,15]. For example, we may wish to determine the value of the generalized stress intensity factor A [see eq.'s (11a,b), (12a,b,c)]. Such procedures based on [13,14,15] have been implemented in *PROBE* [9].

4. CONCLUSIONS AND RECOMMENDATIONS

- (1) Extensions are essential for both the estimation and control of error in finite-element computations.
- (2) We are in a much better position today than we were, even just one year ago, from the point of view of understanding how an advanced finite element software system should be designed so that (a) the solution is obtained at very nearly the theoretically optimal efficiency and (b) the user is provided with the capability to estimate and control the quality of engineering data computed from the finite-element solution at a small marginal cost. This is because now we understand the interplay between mesh design and the polynomial degree of elements.
- (3) P-extension, coupled with properly graded meshes, is the most efficient method for controlling error in finite-element computations.
- (4) The proper mesh design is such that points of singularity (and areas where the solution changes rapidly over short distances) are isolated by one or more layers of small elements, with the elements graded in geometric progression toward the points of singularity. In this way both the global and local behavior of the solution can be represented without compromising the accuracy of either.
- (5) Implementation of advanced extraction methods for the the computation of certain engineering data, such as stress intensity factors, will further increase the efficiency and reliability of computations.
- (6) The p-version is well suited for implementation on parallel processors because the data are organized in relatively few, large units. This logical organization reduces the overhead associated with parallel processing.
- (7) The substantial increase of efficiency in finite-element computations through the use of h-p extension and the availability of parallel and vector processing technology make it possible and desirable to model plate and shell problems

using hierarchic sequences of plate and shell theories in conjunction with fully three-dimensional representation. The various plate and shell theories are nothing more than specializations of the three-dimensional theory of elasticity through restrictions imposed on the variation of the displacement field in the direction of the normal. Such restrictions generally do not hold near supports, stiffeners, cut-outs, plate and shell intersections, etc. which are the areas where cracking and delaminations originate and therefore of the greatest concern to analysts and designers. These areas can be properly modeled by three-dimensional representation only. The use of hierarchic extensions toward higher order plate and shell theories will permit us to assess and control the quality of approximation in relation to three-dimensional theory.

- (8) Although linear theory is properly the first and most generally used approach to structural modeling, it should be possible to ascertain by a posteriori analysis whether engineering conclusions drawn from a numerical model would be different if geometric and material nonlinearities were considered. We can view linear theory as the simplest of a hierarchic system of theories. Much the same way as we estimate error by the use of extension processes within the framework of linear theory, we should be able to estimate error by extension within the hierarchic system of theories. This important area has not received much attention in the past. Because it bears on the reliability of computed data, and the engineering conclusions based on them, it deserves serious consideration.
- (9) In some areas our ability to compute data is already greater than the material scientists' ability to tell us what data should be computed. For example, it is not fully understood what parameters govern crack initiation. The reason, at least in part, is that the conventional finite-element method tends to yield 'fuzzy' data in areas where stresses change substantially over short distances. Proper use of h-p extension, coupled with advanced extraction methods, permits us to compute any stress field parameter with arbitrary precision. This removes an uncertainty from the phenomenological characterization of material response to various stress fields. Of course, such characterization can be developed only through joint experimental-analytical investigations.

5. REFERENCES

- [1] Babuška, I., Szabó, B. and Katz, I. N., "The p-Version of the Finite-element Method", *SIAM J. Numer. Anal.*, Vol. 18, pp. 515-545, (1981).
- [2] Babuška, I. and Szabó, B., "On the Rates of Convergence of the Finite-element Method", *Int. J. Num. Meth. Engng.*, Vol. 18, pp. 323-341 (1982).
- [3] Babuška, I., Gui, W. and Szabó, B. A., "Performance of the h, p and h-p Versions of the Finite-element Method", Institute for Physical Science and Technology, Laboratory for Numerical Analysis, Technical Note BN-1027 (September, 1984).
- [4] Gui, W., "The h-p Version of the Finite-element Method for One Dimensional Problem", Doctoral Dissertation, University of Maryland, (1985).
- [5] Guo, B., "The h-p Version of the Finite-element Method in Two Dimensions", - Mathematical Theory and Computational Experience", Doctoral Dissertation, University of Maryland (1985).
- [6] Szabó, B. A. and Babuška, I., "Stress Approximations by the h- and p-Versions of the Finite-element Method" *Proc.*, 6th Invitational Symposium on the Uni-

- fication of Finite-elements, Finite Differences and Calculus of Variations", Edited by H. Kardestuncer, The University of Connecticut, May 1982.
- [7] Szabó, B., "Estimation and Control of Error Based on P-Convergence" Proc. Int. Conf. on Accuracy Estimates and Adaptive Refinements in Finite-element Computations (ARFEC), Lisbon, Portugal (1984).
 - [8] Williams, M. L., "Stress Singularities Resulting from Various Boundary Conditions in Angular Corners of Plates in Extension", J. Appl. Mech. ASME, pp. 526-528 (1952).
 - [9] Szabó, B. A., "PROBE: Theoretical Manual", Noetic Technologies Corp., St. Louis, Missouri (1985).
 - [10] Szabó, B. A., "Mesh Design for the p-Version of the Finite-element Method", Report WU/CCM-85/2, Center for Computational Mechanics, Washington University (1985). Presented at the Symposium on Recent Advances in Computational Mechanics, Joint ASME/ASCE Mechanics Conference, Albuquerque, NM (June 24-26, 1985).
 - [11] Szabó, B. A., "Implementation of a Finite-element Software System with h- and p-Extension Capabilities", Proc., 8th Invitational UFEM Symposium: Unification of Finite Element Software Systems, Edited by H. Kardestuncer, The University of Connecticut, May 1985.
 - [12] Szabó, B. A., "Computation of Stress Field Parameters in Areas of Steep Stress Gradients", Communications in Applied Numerical Methods, Vol. 2, pp. 133-137 (1986).
 - [13] Babuška, I. and Miller, A., "The Post-Processing Approach in the Finite-element Method" - Part 1: Calculation of Displacements, Stresses and Other Higher Derivatives of the Displacements", Int. J. Num. Meth. Engng., Vol. 20, pp. 1085-1109 (1984).
 - [14] Babuška, I. and Miller, A., "The Post-Processing Approach in the Finite-element Method" - Part 2: The Calculation of Stress Intensity Factors", Int. J. Num. Meth. Engng., Vol. 20, pp. 1111-1129 (1984).
 - [15] Babuška, I. and Miller, A., "The Post-Processing Approach in the Finite-element Method" - Part 3: A-Posteriori Error Estimates and Adaptive Mesh Selection", Int. J. Num. Meth. Engng., Vol. 20, pp. 2311-2324 (1984).

Wednesday, June 19, 1985
LOCAL/GLOBAL NONLINEAR STRESS ANALYSIS

Questions and Answers following: "Some Issues in Numerical Simulation of Nonlinear Structural Response" by H. D. Hibbitt.

Nelson Bauld, Jr., Clemson University: You used a linear constraint, a Riks method, for computing the limit point on the buckling load. Have you tried the Crisfield method?

Hibbitt: We played around with a number of these methods, and Crisfield has done quite a lot of work on Riks' algorithms. We've tried some of the things he's mentioned. I think everybody has his own little perturbation or variation on the Riks concept. We're using one that I found out a few months ago is exactly the same as Issac Fried published. We're pretty happy with it, in most cases. It certainly has done as well as the alternates we've tried. The linear constraint makes it very easy to code and deal with in a program like ABAQUS. It's critical to use some kind of automatic incrementation scheme along with it. By and large, it seems to work well. In the problems where it fails, I've certainly had failures with some of Crisfield's alternate algorithms as well.

Questions and Answers following: "Computerized Structural Mechanics for the 1990's" by B. F. Backman.

W. J. Stroud, NASA Langley Research Center: Bjorn, in your talk, you mentioned computer program development and the usefulness, most often the lack of usefulness, of these computer programs to industry people. That is really a problem. I'll give you my personal opinion about some of these things. It is very difficult for NASA to develop computer programs for industry. As a result, except in some very special cases throughout our history, we really have not had that as our goal. On the other hand, as a part of our research, we and our grantees and contractors develop computer programs because that is how we calculate numbers. We try to make those codes and the supporting technology available to industry. But it's very hard, as you well know, to develop computer programs for someone else's use. Would you like to comment on that?

Backman: I think NASA has a key role in this area. When I referred to second generation programs I meant that there is a strong need for an organized interface between the developers and the users, and I see NASA as having a key role here. Maybe it's a standing commission--maybe it's an established relation for review--I don't know. But where the needs can be established and where the first generation software could be used and evaluated by the industry and recommendations coming back to the developers, I think there exists an interest in this. The only question is how you stimulate this to the point where it becomes mutually beneficial.

Questions and Answers following: "Nonlinear Response of a Blade-Stiffened Graphite-Epoxy Panel with a Discontinuous Stiffener: Work in Progress" by N. F. Knight.

K. C. Park, Lockheed Palo Alto Research Laboratory: Norm, is that specimen based on an optimized design for which the panel buckling load is about the same as the skin buckling load?

Knight: This panel was one that's been around for a number of years. It was first built as part of one of the early studies in our composite design program and, I believe, it was designed without the hole--as part of a wing-type structure near the tip, so it is primarily a strength-type panel. Now with the hole, the characteristics of the panel change.

Yung J. Chiang, Uniroyal, Inc.: By using the point-stress failure criterion, you assume that the failure occurs when the stress at a distant d_0 away from the edge of the hole reaches ultimate. What is the stress that you are referring to?

Knight: We applied just the average inplane stress resultant. It was just an average longitudinal stress-type measure. We were using stress resultants not necessarily to predict failure but to guide us in defining the mesh refinement needed near the hole in order to predict the stress distribution. The elements that we were using do not have a transverse shear formulation, so we can't really recover the shear stress distribution other than from classical plate theory by taking derivatives of the moments and things.

Chiang: Can the energy calculation be better than stress for the prediction of the failure?

Knight: We didn't calculate the energy so I don't really know. You can calculate a local energy norm and use that to predict failure, but we did not make that calculation. I can't answer that.

Questions and Answers following: "Computational Perspectives on Postbuckling of Composite Shells" by G. M. Stanley.

Gerald Goudreau, Lawrence Livermore National Laboratory: That fifth buckling mode that you said was nonphysical, did it vary with the different element types that you've studied?

Stanley: No, not really, the eigenvalues moved around by a few percent here or there. There was actually more of a discrepancy in the linear solution than in the buckling solutions.

Goudreau: Then to what do you attribute the nonphysical basis for that mode?

Stanley: I'm not saying that that mode is wrong. I'm saying that that mode would not occur in practice, because any imperfection would prevent it from occurring.

Jerrold M. Housner, NASA Langley Research Center: Gary, I noticed that it's a pretty rough problem once you get to the buckling load. In the slide showing experimental results, it looks like there was a distinct softening of the panel just prior to its first failure--collapse.

Stanley: Before you continue, that distinction was a little sharper in this picture than it was in practice. That was handdrawn.

Housner: But, in any case, the analysis doesn't seem to pick that up. The softening is in the last 10 percent of the buckling load as you're coming up the load deflection curve. Could that be--

Stanley: The analysis doesn't pick up the peak; the analysis does roll over.

Housner: It does roll over for a short way I noticed, but what I'm driving at is that the softening that takes place--could that be what triggers the failure and is not being duplicated in the analysis?

Stanley: The slide which shows stress contours in color would help us here. As we move up the load-shortening curve--near, but before, the peak--the analysis predicts very high (near ultimate) axial stresses in the vicinity of the hole. So, it's very possible that delamination is already occurring up there. This softening is due to material instability. (See figure, p. 207).* These are the compressive axial stresses in the, I think, fourth layer--in which the fibers are oriented along the axis--and this color scale is such that the maximum is white and the minimum is black. Everything in white here, though, is actually beyond the nominal ultimate stress for this particular composite. So at the buckling load we're already at ultimate stress on both sides of the hole. This shows that as we roll around, the white region begins to rotate a little bit, then is unloading and drops off. When we're back on the secondary path, all the load is picked up here and it's again way beyond ultimate. You can't read it from the scale, but as we go up the curve, it's maybe 50 percent or more higher than ultimate. So there are a lot of things that need to be explained, and I think that we need to try one thing at a time. I don't think we should jump into a 3-D analysis until we have tried some failure criteria--just degrading the material properties.

Housner: That's what your next step is going to be--putting in failure criteria?

Stanley: I think so. I plan to use this workshop, the feedback and opinions expressed here, to help formulate the next phase of the research, as a matter of fact. But, that's one possible direction that we might begin next.

Questions and Answers following: "A Review of Some Problems in Global/Local Stress Analysis" by R. B. Nelson.

Stroud: In the model that Paolo Roberti generated, what was the logic behind the model generation and refinement?

Nelson: In words, the approach is to say: I'll take a specific grid and then I'll refine it one time to see if the stresses change much. And I compare

stresses across neighboring elements. If the stress change is more than a set tolerance (and the stress change was, by the way, a root mean square type of stress change), then I will refine the model further in these particular areas. There are a number of issues regarding the convergence that should be used and how one should go about this. There are a number of authors who have written on that topic. I'm only bringing it forward to show that it is an area which I think merits serious research consideration.

Stroud: I agree with you. I think that many of the things that you alluded to are nice to have. Yet, today, it is difficult for us to get support to do some things that might even suggest having an expert system built into it because that's considered by many to be a diversion. When you have a limited amount of energy and funds to put into something, you can't go into these diversions.

Nelson: It is true, but I think that the development of an automated technique for model regeneration, or adaption, or refinement, something that the engineer really shouldn't have to do, is appropriate. The engineer should not have to tend the screens. So many people tend the screens, and I think they ought to be putting their time to exploring whether or not we need to have a large deformation option being turned on, or whether or not we are worried about there being a problem with the material model used to model a composite. Is it a mean square of stress or a delamination stress that we should be looking at? Those kinds of issues are engineering issues I think should be pursued.

Questions and Answers following: "Some Comments on Global/Local Analysis" by S. N. Atluri.

James C. Robinson, NASA Langley Research Center: I think when we start talking about either globally or locally exact solutions, as structural engineers, we should show proper respect for a negative force displacement curve. It looks very nice when it's plotted up in a rigid test machine, but when it is in an actual structure that is surrounded by other equally sensitive elements and you don't do a dynamic solution, you have probably missed failure at the ultimate load point. To look at it from there on is a fine academic exercise, but structurally useless.

Atluri: I agree with you totally. These are illustrative examples to test the theory that is presented. In these examples, a displacement-controlled loading is assumed.

K. N. Shivakumar, AS&M, Inc., Hampton. You mentioned that in a composite panel, some theories would allow you to replace a hole by a crack. What are the theories? To me, a crack has an entirely different model from a hole, and you may be missing some physics in the problem.

Atluri: If you consider an isotropic plate with a hole in it, the stress concentration is always three, if the size of the plate is roughly five times bigger than the hole size. But in a composite laminate, there is a size dependence. Even if you consider a plate five times bigger than the hole diameter, the stress concentration still depends upon the size of the hole. And in that sense, if you were to plot the failure stress of the laminated plate with the hole versus the hole radius, it would vary as r^α . And in fracture mechanics, that variation is--if you were to consider r to be the length of the crack--the failure stress for a cracked plate would vary as $r^{-.5}$, or something like that. There is no great theory behind it--just an empirical relation. Mar at MIT has taken that, and there is a theory called the Mar-Lin theory wherein the fracture stress in a laminated hole varies as the diameter of the hole to some power alpha. So there is some evidence that a hole in a laminate behaves like a crack. The physics of that needs to be explored much further.

Stanley: I agree with your advice to incorporate locally exact solutions in global analysis, but in this particular case of the composite panel with a hole, what about the subsequent propagation of failure in which the hole begins to distort--it's no longer a circular hole. Do you think it's actually feasible to continue to follow the complete analysis through with locally exact solutions?

Atluri: I think so. At least these locally exact solutions could give you a better handle on the three-dimensional stress state--like the transverse shear and so forth. And when large geometric changes take place, those solutions would not be locally exact, but they would be much better than the traditional finite element basis functions you use.

Stanley: Perhaps they could be incorporated as deviational basis functions on top of the normal shape functions.

Questions and Answers following: "On Computational Schemes for Global/Local Stress Analysis" by J. N. Reddy.

Glenn Sahrman, PRC Kentron: In your analysis of the 2-D free edge problem, were the stresses becoming singular--unbounded?

Reddy: Without free edge cap, yes, the analyses done by many people show that as you refine the mesh, σ_z still keeps getting larger. So, it is unbounded. But we know that σ_z must have some finite value. I did not address that problem here. My objective was is to reduce the stress concentration by adding a cap. We accounted for the cap in the finite element analysis. We also modeled the cap as part of the region.

Questions and Answers following: "Global/Local Finite Element Analysis of Localized Stresses in Prismatic Structures" by S. B. Dong.

Housner: After you compute what you feel is the position of the interface between the local and the far field solutions, have you gone back to examine the sensitivity of the solution to the location of that boundary--that interface?

Dong: Yes, in fact, I had a student that took the interface at different regions. What happens is that when you have self-equilibrated stresses, those which you subtract out from what the Bernoulli-Euler theory or Kirchoff theory would give you in terms of a plate analysis in the far field, the self-equilibrated stresses will decay. If you take the finite element region very close to the localized area, you would obviously need more global functions. But that's really not a severe penalty because when you include 10 global functions, you can probably eliminate one or two layers of finite elements that would involve many more degrees of freedom. A second advantage of taking the finite element region very close to the localized area is that you will determine on a mathematical basis how the self-equilibrated stresses diffuse as they go into the interior rather than by letting the finite element solution--which depends on the modeling--indicate how the self-equilibrated stresses are diffused. If you have a complete set of global functions, then there's no difficulty in going as close as you please to the localized region. By successively taking more and more global functions, you have a basis for evaluating the error that's incurred in the analysis.

Housner: I guess you could go all the way back to the inclusion itself if you really wanted to. Years ago, I dabbled a little in wave propagation and if you have the complete set of waves, including those that attenuate, you can actually, theoretically, at least, go all the way back to the inclusion. Although, in those days years ago, we didn't have the computers to actually do that.

Dong: I think if you go back to the inclusion, Eli Sternberg would be very happy. Then you would cut out all the finite elements, and you would be where he was before all of us messed up his conquest of all the problems we have.

Questions and Answers following: "Global/Local Methodologies and Their Applications to Nonlinear Problems" by A. K. Noor.

Park: What was the motivation for choosing a symmetric boundary condition? We have found that a symmetric boundary condition does not yield the physically correct buckling solution.

Noor: Well, I think you have to qualify the type of symmetry. If you say reflection symmetry or mirror symmetry, then I would agree with you. What we found in these problems is that you have inversion symmetry which is typical of the response of many anisotropic panels. I have looked at the results that were presented by Norm Knight and Jim Starnes and we also looked at our results. The inversion symmetry was exhibited by the response of the panel up to the maximum load given by the experiments.

Park: Do you foresee that one may choose this boundary condition to do collapse and also postbuckling analysis?

Noor: Let me put it this way. If you start from the beginning conducting a nonlinear analysis, the inversion symmetry would always be preserved until and unless you have a branching point. And you can always detect the branching point from your equations. If you get that branching point, then there is a possibility that you might go on a branch where that symmetry is lost. However, as you will hear tomorrow, there are ways then to synthesize the solution of the unsymmetric problem from the symmetric and antisymmetric solutions, even for the

nonlinear case. Summarizing what I said, I think if you start from the beginning with a nonlinear solution, the inversion symmetry properties of that nonlinear solution will not change unless you have a branching point.

Park: How far have you carried along your solution in your example?

Noor: In the unstiffened panel, up to the maximum load that was given in the experiments.

Park: We happen to disagree with that because just before the collapse there is a bifurcation that occurs about 10 or 15 percent below the collapse load, and that mode happens to be antisymmetric.

Noor: We did not detect bifurcation up to the level of the loading that we had, but if you have bifurcation, you can lose the symmetry. But even then, as I said, you can synthesize the unsymmetric solution from symmetric and antisymmetric components. And you'll hear about this tomorrow.

Stanley: I think it would be good for us to compare quantitative results, but just qualitatively, are the path derivatives more accurate than the modes in this particular problem or has this been your general experience?

Noor: We did not try the buckling modes in this particular problem but I tried them in several other problems--plate problems, shear-loaded plates, axially-loaded plates--and without any exception, you get considerably higher accuracy with the path derivatives than with equal number of buckling modes. Not to mention also that the path derivatives are considerably less expensive to generate than the buckling modes because if you generate them at zero loading or zero end displacement, you need to decompose only the linear global stiffness matrix.

Questions and Answers following: "Application of the p-version of the Finite Element Method to Global/Local Problems" by B. A. Szabo.

Carlos A. Felippa, Lockheed Palo Alto Research Laboratory: I think your theory depends essentially on the assumption that the elements are conforming.

Szabo: Yes.

Felippa: Now, most production computer programs do not use conforming elements. And if you use nonconforming elements, a funny thing can happen. You can refine the mesh and get a worse solution because the aspect ratio of the elements may deteriorate. How would you extend the theory to programs that use nonconforming elements?

Szabo: To me, a very important consideration is that there is an existing proof that the solution will converge in a certain norm. There are several methods, such as methods involving nonconforming elements, which do not meet with the Babuska-Brezzi criterion which guarantees the desired convergence properties. This criterion is difficult for engineers to understand because it is based on mathematical considerations. There are several methods in existence (methods that do not meet the Babuska-Brezzi criterion) for which one could select a set of input data and the methods would work well. In other words, one could design the mesh, select the input data, etc., and the method would work. And somebody else could select a different set of input data, and because the method does not satisfy the Babuska-Brezzi condition, it may "blow up". Now it frequently happens that a new method is shown to work on smooth problems, typically textbook problems. And the researcher says, "Look, my method works because it can solve some specific problem like the simply supported square plate." Any numerical method should be able to do this. The real difficulty is when these methods are applied to nonsmooth problems which are the important problems in engineering. Then they may not work; they may give a completely nonsensical result. For this reason, I choose to stay away from those methods. Instead, I use a method that is based on the principle of virtual work because I know this method satisfies the Babuska-Brezzi condition. Nonconforming methods, which do not satisfy those conditions, may give you unpleasant surprises. For example, you may be doing more work and getting worse results. On the other hand, if you use a method that satisfies the Babuska-Brezzi condition and if you do more work, (for example, use finer meshes) the analysis will give you better results.

Stroud: Because of Carlos' comments and some comments that you just made, do you recommend that conforming elements be used exclusively?

Szabo: Yes, until proofs are developed that the other methods are robust--in the sense that no matter what (admissible) data you select, the method should work. And this proof is not available for the nonconforming elements, as far

as I know. Now it could be that somebody has just worked it out yesterday (or something like that), but I have talked with mathematicians about this and they have the view that it is dangerous to use methods which are not meeting that criterion.

Reddy: I think that the LBB condition is not really a necessary condition, it's a sufficient condition. And I think that there are some elements which are proven to work even if they do not satisfy the LBB condition. I think that we should not just throw out everything that will not pass the LBB condition because it is only a sufficient condition. Not necessary.

Szabo: I've heard that argument. I'm not a mathematician; I've only picked up information on this topic by osmosis. But I believe that it is a necessary condition. Basically, the argument is the following. If your method does not meet the Babuska-Brezzi condition, then one can always select input data (maybe not the kind of data that you are interested in) for which the method will not work, and for which you were not able to say ahead of time, "Please do not make my input data like this or that," because it is very difficult to predict which set of input data will cause the method to fail. But please don't press me on issues that relate to finer mathematical points; I'm not a mathematician.

Reddy: I'm only trying to let the other people here know that if they check the LBB condition and if it fails then they should not give up on the method. The same thing about Lax-Milgram theorem for existence. It is a sufficiency theorem, but not necessary. It can be shown that a problem has a solution but may not meet the same conditions as stated in the Lax-Milgram theorem or Babuska generalized theorem.

Szabo: I just would like to invert your question. If you know you have a good method, for example the method based on the principle of virtual work, why would you wish to select a different method which may give you unpleasant surprises? In other words, since we do have good methods that we know are working, what would be the reason for using methods--nonconforming methods, for example--that have not given satisfactory solutions at various times?

Reddy: Well, very rarely do we use methods other than those based on virtual work principle. For example, mixed-type finite elements are used in certain problems where we think stress is an important quantity and that should be known

on the boundary of the element, like hybrid methods and mixed methods. Those are the cases where you need to use mixed methods which are not based on the virtual work principles--I should say traditional virtual work principles because the mixed principles are also some kind of virtual work principles.

Szabo: I'm not against any particular method--or for it--all I am saying is that I would like to see a set of objective performance measures, convergence characteristics in certain norms, proven and demonstrated, in order for me to be convinced I should use that method. Until that is available, I don't want to use it. And I have not seen really detailed performance studies on the methods that you have mentioned. It could be that they exist. All I'm saying is that I haven't seen them and I'm quite satisfied with the performance of virtual work formulations.

Reddy: I think that history shows it works the other way around. We, as engineers, design methods which we think indeed do work. Then mathematicians, or people interested in mathematical aspects, explore the question of whether it is really a right method.

Szabo: I agree. If we waited for the mathematicians, I think we would still be living in caves. But, on the other hand, a large part of this meeting is about concerns of the performance of the various methods, and that's what I wanted to address. I would love to see similar performance studies presented for the other methods as well. I haven't so far seen those.

Panel Discussion

Bjorn F. Backman, Boeing Military Airplane Company: I would like to start by making a controversial statement. I think any product of engineering development in the methods field is an engineering tool. And I would like to make the distinction here, initially, between scientific development and engineering development. I think science is involved in the pursuit of the truth, while engineering is involved in the rational process of making decisions in an environment of limited knowledge. To that extent, I would have liked to see one of the two things. I would have liked the end product of computational structural mechanics to either be a tool that directly applied to the design process or a tool that comes with a recipe that says this is how you use it in order to prepare the set of data to be used for the design process. That's from

one side. On the other side I see if we want to--in an academic environment--to pursue the scientific side of engineering to the exclusion of design considerations, then I think NASA has an even more important role in making the translation into the tools that can be applied to the design process. Thank you.

W. Jefferson Stroud, NASA Langley Research Center: Yes, that was a controversial statement. I want to make sure I understand what you said, Bjorn. You feel that the CSM activity should produce products. I will even go further and say software products.

Backman: Yes.

Stroud: Would you say that merely developing the know-how to produce the software would be a shortfall?

Backman: Well, it reminds me of the difference between solving problems in principle and solving problems. If you look at any solution technique that's going to require you 3 months to come up with a number. If you want to apply that technique to the design process where the question is not what is the response, but the question is, what is a good design--(For instance, for this curved panel that we have been studying, where is the optimum location of the hole, or what is the best pad-up around the hole, or what is the sensitivity of the design to realistic boundary conditions?)--then you see 3 months very quickly translates into at least a hundred times that long in order to solve the design problem. So I'm implying that if you don't ask yourself the question, what am I going to use this technique for, or you say this is the way you can use it in the design process, you have only solved the principle not the problem.

Gary M. Stanley, Lockheed Palo Alto Research Laboratory: I'd just like to comment. I think that what he's saying is that the implementational aspect of methods development is nontrivial and that we should be pursuing methods implementation issues simultaneously with methods development--rather than waiting till later.

David Herting, MacNeal-Schwendler Corporation: We're sort of coming in from left field here. Let me cut in, because Universities and NASA have been

dancing this nice little dance, and we are involved in this in several aspects. We (at MSC) are involved in this research ourselves, and we are also involved in developing similar code. We are now out in the business of paying for code development ourselves, in effect, also doing what NASA's doing. So this is all very interesting to us. However, I'd probably like to make a couple of comments.

One thing I'm glad to see is NASA is getting out of the role of trying to develop massive computer codes. I think that's a good trend. If you go into the research mode it's going to be much more valuable to us, and much less duplication. Also, I'm glad to see that you have practical applications. I'm glad to see people looking at the realistic problems and developing techniques to solve them because we are also looking at the same problems, particularly tires and cracks. We are also in the development mode.

However, I'm disturbed about a few things. I've kept quiet here during the morning, and maybe I should bring up a couple of points that should stimulate some discussion. One of the things is the lack of knowledge about the commercial codes. I speak for Swanson, I speak for Hibbit. A lot of the complaints that were coming up about the codes are not true. A lot of these capabilities exist. The capability to do the localized nonlinear analysis is implicitly in NASTRAN, or MSC NASTRAN anyway. I'm sure ABAQUS has similar capabilities.

This brings up the previous point of the ivory tower aspect. It appears that you are in an ivory tower if you're not aware of what's in the codes. I think it should be a part of the academic world to look at commercial codes. We certainly look at the academic world as far as looking at what the research is and what the current applications and hot methods are. Let's do it a little more equally.

A couple of other comments: Some of these research projects are going to develop new methods, new codes. What's going to happen to these after they're done? There seems to be a lot of development at NASA that just sort of goes away and hides. We can't seem to find it. Sometimes we find it at COSMIC and COSMIC says, "No, that's our property," and we can't use it. If you're going to develop all these nice element routines--solvers, nonlinear methods--how do we

get those out to the public? We'd certainly like to use them. There may be some restrictions; we'd be glad to work around them.

Stroud: You said a couple of things that I didn't quite understand. Although COSMIC charges for the software, I thought you could get any software that you wanted.

Herting: Well, the question occurs because we'd like to put it in NASTRAN and distribute it out to everybody in the world.

Stroud: Worldwide distribution of COSMIC software is different. If software is developed with government funds, and if it's considered to the advantage of the United States to have the software available only to United States companies, then there is a restriction. But as far as COSMIC restricting software from anybody in the United States, I don't believe that's the case.

Herting: We've had some troubles just for the United States. I think the problem is, do we go out and require every customer to pay for a COSMIC code that is incorporated in MSC/NASTRAN? Take a program like CONMIN that takes \$200 or \$300 to purchase from COSMIC, our legal staff tells us that we have to go collect that \$200 or \$300 from every delivery we make of CONMIN and ship it off to COSMIC. And it's really questions--details--like this that we haven't been able to work out.

Stroud: I can't address that question. Your other question was something that I was really concerned about when we were considering having software developers at this workshop. Would the software developers say, "We can already do all these things." I think that a sharp person with a good existing code can do a lot of things. I would not question that one bit. A sharp person with a good existing commercial code could do all of the local/global stuff that we've done. But I think you agree that it would not be easy or routine. We want to begin now to look ahead at what we ought to be able to do 5 to 10 years from now.

H. David Hibbitt, Hibbitt, Karlsson, and Sorenson, Inc.: I'd like to address the comment that the academics should look at commercial codes. I didn't come here to be looked at, I came here to look. We commercial code vendors can learn a lot from academic work. Our job is to try and package it to make it sensible for the guys out there who have real problems. I don't really think it's

terribly important that the academics look at us. I came here, in fact, to look at an issue that is becoming important and that I haven't heard discussed very much except in very general terms. And this is the issue of parallel machines. Langley, I think, is the place where you have the finite element machine. On the other extreme, people at Cray tell me that the way to do parallel processing is to get the data down into a microlevel and do a lot of local parallelism. I hope our code is going to have a long life. So we really have to look at these parallel architectures and we have to decide what to do. Now I haven't heard anybody here discussing the finite element machine; I haven't heard anybody addressing detailed issues of writing software on parallel machines. Is there somebody here who can help me with that, and where is he, and could he address the question?

Stroud: You are right, Dave. We are doing work in parallel processing. And there is no paper here today that is oriented specifically towards parallel processing. Tomorrow, Joe Padovan will talk about it. He's into tire modeling. We are hoping that within the next year or so we will be able to have a conference that will focus on parallel processing in structural analysis. We're trying to learn. We're buying a new multiple processor computer that will replace the finite element machine for parallel processing research. It's called a FLEX/32 MultiComputer. Each one of the 20 processors is about like a VAX 11/750. And I hope that we will be able to address some questions that you raised. Why don't you throw the question of parallel processing to the panel and audience right now.

Hibbitt: I think that the point that I was getting to, Jeff, if I could just continue for a minute is it seems that the philosophy you've taken here at Langley is to go into parallelism at the macro level, putting one finite element on one processor, while if you talk to people, say at Cray Research, the impression I get is that they want to do it all on a micro level. There are some very successful finite element applications that have taken advantage of this approach. I think Jerry Goudreau is here, and I think he knows a lot about this. And I was hoping that he would give us some good advice here. Is there a right and a wrong way, or are both ways successful?

Stroud: Let me comment on what our group has been doing in parallel processing. First of all, the Finite Element Machine started out, in the mid-to-late 1970's, with the concept of a microprocessor for each finite element. By the beginning

of the 80's, we decided that one processor for each finite element was not the way to go. We just got 16 processors operational on the Finite Element Machine about a month ago. Granted it was not a high priority research item, but it was clear that we here at Langley could not develop a computer having several hundred processors. So we and several university grantees have been looking at algorithms that could exploit multiple processors. For example, we are looking at linear simultaneous equation solvers--both iterative and direct solution techniques. Another example is eigenvalue solution techniques in which you have eigenvalue shifts. You could assign a value of the shift to each of the processors and let each processor get the eigenvalues near its shift parameter. You could use some logic to insure that you've gotten all the eigenvalues. A third example is substructuring, in which you divide the structure into pieces, assign those pieces to different processors, let each processor calculate the response of the substructure assigned to it, and then work the interface problem. In each case, computational speedups can occur if several processors are working on their assigned tasks simultaneously. But, the approach that we've totally abandoned is the idea of a microprocessor per finite element.

Gerald Goudreau, Lawrence Livermore National Laboratory: Let me just take a moment, not to preempt the panel, but to respond to Dave a little bit about our experience in a Cray environment with primarily the pipeline concept of the Cray. We also have implicit codes that concern themselves with optimization of profile equation solvers and eigen packages, but a large portion of our finite element applications is in dynamic impact using explicit finite element hydra codes which don't have any matrices to deal with whatsoever. And so without linear algebra to exploit, what are you going to do to take advantage of the Cray environment? And the thing that we hit on, and this is John Hallquist, and primarily in the DYNA code applications, is that a big chunk of the effort, in fact 95 percent of the effort, is on what you would call the right hand side evaluation, you know, f of x and t , which is really f of the stress state. It's the element stress divergence in solid mechanics, and there are comparable things (B -transpose σ) in any of the structural elements as well. And rather than try to vectorize the element, meaning that the loop over whatever degrees of freedom are within an element, we used element chunking. In element chunking, first you scalarize every aspect of the algorithm within the element and then, in the innermost do-loop, you tightly code a loop over a large number of elements. Now some of the vector machines require very large elements, and if you turn all the scalar variables that are in the element logic into vectors

over a set of elements, you may have a horrendous amount of storage required to get long enough vectors. But with both 64 and 128 vector lengths and with the memory that the Crays have provided, we've been able to chunk at that level very very efficiently. And I think the B-transpose sigma part that--and also the strain computation--they're just about at the limit of what we think we can get out of it--the Cray environment that we live in. This does present a problem with constitutive models, or equations of state because every time a new one comes along it immediately slows down the code by a factor of 2 to 10. It's only after it's proven its worth that we then get in and vectorize or optimize that element.

Now let me say a word about parallel machines which is sort of concatenating the pipeline to do more. The Cray XMP and future Cray machines are promising us more, and we have John Hallquist's protege here, Dave Benson, who has been itching to multitask the XMP. A problem we have at Livermore is that when the XMP is not in the hands of one research group, but in the hands of a laboratory with 500 to 1000 equally important researchers, what you find is that the multitasking priority is not for individual code, but for the operating system as a whole. And so at Livermore the operating system is trying to multitask all these users trying to simultaneously do their work. And so, at the moment, all we have access to is a single processor and we haven't gotten advantage of that. Now I was up at White Oak, the Naval Weapons Center, yesterday and found out that ABAQUS is being explored by CSPI, a small array processing company up in Massachusetts, and we're getting one too I guess, not in my group, but another group at the Laboratory. So I would be interested in talking further with you about--maybe at a smaller level than at a Cray--looking at some of these array processing ideas and how we could proceed and then maybe influence the bigger machines down the road.

Moktar Salama, Jet Propulsion Laboratory: I was also hoping to hear from the speakers more about how their problems fit into the broad objective of the CSM activity, which as stated here, is to develop advanced structural analysis technology that will exploit modern and emerging computers. I was hoping that the speakers would give us an idea of the amount of computing that their problems take. Because if the problems we're talking about here require little effort, little computing, then today's computers are adequate. Unless there is a great deal of computational intensity, a large amount of data, or unless the kind of problem that's being solved requires a real time solution then these problems

don't fit into the category of problems that need modern computing beyond what we have today.

Barna A. Szabo, Washington University: I had occasion to speak to many aerospace engineers in different companies. Their general opinion is that as the 1990's come along they foresee the use of 3-dimensional finite element technology in airframe design. The wing skins are getting thicker and more complicated, and new materials are being used. There is no doubt in their minds that the aerospace industry will be directed toward very heavy computing, very large computational loads, very complicated geometries, many elements, new materials, singularities, damage tolerance requirements, and the like. As we look to the 1990's, I think it is clear that parallel processing is going to be increasingly important. And I believe they are very interested in the technology I was addressing--where we can obtain high accuracy with much fewer degrees of freedom by properly designing the finite element spaces. There is a real need out there for us to look at how the airframes for the 1990's will be designed.

On the other hand, I cannot really speak about parallel systems because such systems are not accessible to us at the university. We're very happy to find time on the VAX 11/750. Even there I have to twist the dean's arm. I would certainly be very interested in exploring different computer systems to find out exactly how we should design algorithms for those systems. Unless we have access to parallel systems to experiment with them--I don't think we can really answer your questions and Dr. Hibbitt's questions, which are very important questions. It's not that we're not interested in parallel processing. It's just that we first have to get access to them.

Salama: Well, I'm really not saying that the problems you discussed do not require heavy computing. I'm not saying that. I'm saying that this fact was not brought out, that's all.

Stroud: The CSM group is going to be having grants with universities. We are going to make our Flex multiprocessor computer available to the universities that we are working with. We hope that 12 to 18 months from now we ought to be able to have a handle on several structural analysis methods that look good on multiple processor computers.

Richard B. Nelson, U. C. L. A.: I want to second the comment that Professor Szabo made, and that is that UCLA recently got some new computing equipment

which did involve an array processor. And now we are all looking around, saying to ourselves, now that we have it what are we going to do. Money is the mother's milk of research and for us to go out and explore array processing in a big way we need to have support for a relatively intensive program. With Ph.D. students, one of the things we do is invest a great deal of time in having a student generally write, almost from scratch, a finite element code, or pick up SAP 4, which is by now quite dog-eared, or possibly NONSAP, one of the codes which we more or less have inherited from Berkeley more than 10 years ago. I'm speaking as a researcher in need of a kind of test bed that we across the country could pretty much rely on and plug our research efforts into so that people across the country could make use of our research product on a general basis. I don't consider that to be particularly competitive with optimized codes that are specifically designed for certain industrial applications. But rather that it gives people both in industry and in the academic community the opportunity to communicate across technical lines, and let's face it, the language of communication today is through computers and software. And we have to develop some kind of a liaison through channels of software and that's going to require significant support to develop.

E. Thomas Moyer, Jr., The George Washington University: I'd like to make two comments, one on the hardware aspect of parallel processing. As the technology gets cheap enough, rather than making time available at one central location, it's becoming feasible for universities to buy parallel processing computers. For instance, with the new INTEL machines, you can pick up a parallel system with 32 processors in it for under \$100,000. As the prices fall, universities should buy the computers either through a NASA program or an NSF program in a wide way so that a lot of researchers will have access to them. As far as the comment that was made with regard to university professors not being aware of finite element codes capabilities and that type of thing, I think that the universities should have a lot stronger access to the codes at either the code level or, at least, at the executable level at a price, if not nothing, then extremely negligible. And that I think would be productive both to the commercial endeavor and to the research environment as a whole. And I think that that ought to be explored in more depth.

Stroud: My bet is that commercial software companies do provide their codes to universities at quite reduced rates. (Agreement from audience.)

Backman: I wanted to address the question about the needs of the aerospace industry. I think it's quite clear that for the 1990's we have seen a number of problems, many of which perhaps have not been very dominant before, that require both hardware and software improvements of orders of magnitude. You look at the optimization side, you look at the emerging technology in the damage tolerance fields that--in composites--require delamination growth under compression, and postbuckle states, and I see substantial development requirements. There is very little doubt that there are a number of applications producing problem sizes such that present software and hardware is not adequate. We still can't get into the large optimization problems that we need to be doing. We really need to gain additional first-hand hardware experience in order to truly understand and develop the requirements.

Stroud: We've touched on multiple processor computers. What about the idea that if we are going to have good structural analysis in the future, we need to have better software architecture in order to exploit the new computers.

Leonard A. Lopez, University of Illinois: I think we do need to take a very serious look at the architecture of computer systems for finite element analysis. Dr. Nelson talked about testbed systems. I am a strong believer in that approach. We have been running a testbed system at the University of Illinois since 1975. The system uses a virtual machine concept and emphasizes flexibility, machine independence, easy addition of elements, and easy addition of nonlinear material models. Basically these were areas of interest to individuals in our department in the early 1970's, and this capability has permitted these researchers to try new ideas without needing to deal with all of the other problems associated with finite element technology.

The design of the POLO-FINITE testbed is based on 1970 software and hardware technology. The biggest problem we are facing today is that our expectations have grown significantly, and that the kinds of problems being envisioned by our faculty far exceed the capacity of the combined hardware/software testbed. I think we now must proceed to investigate and exploit the architecture of parallel (multi-processor) computers in order to solve the next generation of problems.

There would be many objectives for a multi-processor testbed system. Clearly one of them must be that the testbed be flexible enough to take into account existing multi-processing architectures, as well as architectures which have not yet been conceived. It will take years to develop such a system and

put it into production. The corresponding hardware technology is changing very rapidly, and it will be necessary to be able to move the testbed to various new architectures as they are designed and implemented. In order to give you an idea about how fast things are changing, I would like to again point to our experience at the University of Illinois. Six months ago we were confined primarily to utilizing a Cyber/175 computer. Today we have two national centers for supercomputers--one based on multiple Cray XMP's and the other based on the CEDAR machine. The CEDAR machine is a research machine based on ALLIANT computers (today). We also have a network of Apollo super-micro-computers which can be used as a multi-processor network; it is also viable for finite element calculations. All of these additions in hardware took place in six months. Additional changes will occur within the next one to two years.

We, as individuals working in finite element software technology, must be careful not to box ourselves in, or exclude the kinds of things which will be happening in the near future. I think we can accomplish this objective by seriously studying and redesigning the architecture of our computer systems, and developing new testbeds that are flexible enough to meet the challenges of the future.

K. C. Park, Lockheed Palo Alto Research Laboratory: You are opening a box of worms. Dr. Bahram Nour-Omid and I have been exploring the Cal. Tech Hypercube for the last 6 months. It took us 2-1/2 months to successfully run a one-dimensional wave propagation problem. And then the word I just got from him on the plane was, after struggling about 2 to 2-1/2 months, we just have been able to run a simple 2-D plate bending problem. What this means is that it is very difficult to implement software on parallel machines, at the moment, mainly because the supporting system utilities aren't there. And it's going to take about 3 to 5 years before we engineers can--in a routine manner--use a parallel machine, either a FLEX machine or a Hypercube, or the like. Right now, for example, the FLEX people insist that they have a FORTRAN compiler. But when we looked at it, it really doesn't have a good solid FORTRAN cross compiler and neither does the Cal. Tech Hypercube. What this means is we have to start all over again using the C language. And imagine the conversion labor involved in rewriting sophisticated finite element programs into the C language. So I think we have 3 to 5 years before we can start to see the routine uses of the parallel machines.

The other comment that I would like to make is that parallel machines are not cheap. What that means is that only wealthy private companies, such as some of the profit-making finite element analysis companies, like Marc, or MacNeal-Schwendler, may be able to afford parallel machines. But in the meantime we want to minimize the time it takes to move from university research products to practical applications--say on a parallel machine. Then the cost of the machine has got to go down, or somebody should provide the funds. Otherwise you will not see a wide dissemination of parallel machines.

Stroud: Today, it's not so much the cost of the multiple processor machines, it's the lack of software. That's right, FLEX might not even have a FORTRAN compiler--and that's a problem. But, I certainly hope it won't be 5 years before we are routinely using the Flex computer. I think that we will be able to begin talking about structural analysis on parallel machines in 18 months.

Vincent Godino, Bolt, Beranek, and Newman: I would like to say a few words about our experience in parallel processing. Under DARPA sponsorship, BBN has developed a parallel processor called the Butterfly. We have a 128-node machine currently operational. The Structural Mechanics department, which I head, has been developing structural mechanics software on this machine including matrix operations, Gauss elimination, etc. Based upon our experience on this particular machine, I must take issue with some of K. C. Park's comments. In terms of ease of implementation, our scientific programmers, with no previous experience on any parallel processors, were able to program several extremely effective algorithms in a short time and with minimum help from people experienced on the machine. The Butterfly is available to DARPA contractors through the ARPA net. Next generation parallel processors consisting of thousands of nodes are currently under development. I guess I would like to agree with Jeff Stroud in that I think that we're a lot closer than five years before these machines are going to be real tools being used to solve structural mechanics problems.

Jimmy Y. L. Ho, Lockheed, Sunnyvale: We need to realize that the material properties and various other properties, such as joint behavior, are statistical quantities. They do not have a precise value.

Szabo: The realities are that we do indeed deal with stochastic differential equations. The loading, the material properties, the thicknesses, and so on are really stochastic variables. I'm aware of some research in this area that's just beginning to address this fact. Monte Carlo methods are out of the question because of the size of practical problems. We cannot perform so many different analyses with different input data. But it is possible to study certain bounding characteristics of the data and reflect those characteristics in our answers in an honest way. And the point raised here could really be a very important topic for NASA to take up as a research study--how to be more realistic in the way we treat computed data. The number is not 10.75, but, rather, it is a stochastic variable. Understanding the bounds within which the answer is valid is very important. Delivering a number doesn't mean very much unless I'm able to state the confidence level I can associate with that number.

Robert Melosh, Duke University: I see four different groups of people here, and that's probably the basis for the conflict. I see a group of analysts who want to be responsive to the demands of their employers and turn out good analyses, and right now, as I see it, the analyst has a very heroic job to turn out analyses that he can depend on. There is another group here represented by program developers who are responsive to the needs of the analysts. They want to supply programs to help the analyst do his job, but they don't have the final responsibility for the analysis. The engineer does. The third group is the group you represent, Jeff, the Langley group who want to do relevant research to help the first two groups. The fourth group includes the university people who want to do relevant research to help the first three groups. I think if you want to center on what you should do as a group, you have to decide who you want to satisfy in terms of setting your priorities. Do you want to satisfy the analysts at that end of the extreme or the researcher at the other end of the extreme? You have to settle on that in order to settle on what your objectives will be for research.

Bahram Nour-Omid, Lockheed Palo Alto Laboratory: There are a couple of issues that have come up in my work on the parallel processor--in particular, the one down in Cal Tech. First of all I'd like to distinguish between parallel processors. The one based on single instruction multi-data, which are Cray and Cyber 205 computers. And the next generation computers which are going to be multi-

instruction, multi-data-stream computers. Now these computers come in a variety of different architectures, some of them are based on message passing between processors, and others are based on a shared memory environment. It is quite possible to write programs--code--and get them to work on these computers. But whether those codes are going to be efficient is a completely different story. One can write a Gauss elimination procedure, probably in a matter of months. However, it turns out that on concurrent processors, which are the multi-instruction, multi-data computers, the time taken for a message to go from one processor to another is equivalent to between 10 and 100 multiplies. Therefore, one has to look at algorithms that utilize the given architecture of the computer. It is pointless to look at algorithms that require a lot of inter-processor communication. One has to look at algorithms which don't do that. In particular, the most promising ones that we have seen are iterative methods and the type which are based on partitioning of a structure into substructures.

That brings up another problem which I think is quite unsolved and that is the mapping of a finite element mesh onto a set of processors. People have done a lot of work on partitioning meshes, regular meshes--and mapping them on the concurrent processors. But once the mesh gets quite general, such as the ones we have seen--finite element mesh of an aircraft--then I think it's very hard to say how to automate the process of mapping those meshes on the computer.

Joop Nagtegaal, MARC Analysis Research Corp.: I would like to make two brief comments on two very different subjects. One is the parallel processing. I believe there are a lot of algorithms already around which lend themselves to parallelizing computations. I know of Gaussian elimination equation solvers which can be parallelized by selfcontinuous substructuring technique. Certainly you can do all element calculations in parallel. What I see is missing at the moment is operating software on these multiprocessor machines which we can program such that they work. That's what I see currently as the bottleneck. But I see that things are changing. So, a little bit of time and I see a lot of things going on. The second comment on a whole different subject is on this uncertainty business, stochastic analysis. I'd like to add that NASA is sponsoring such a project. It's a different branch of NASA, but I'd like to think of NASA as one big organization. And there are certainly some promising things going on. And I would be happy to talk about that with anyone.

Stroud: NASA Lewis is doing that work.

Nagtegaal: And there are several people from Lewis at this workshop.

Stroud: We're all one good team.

Nagtegaal: That's what I like to think, yes, thank you.

Stanley: I'm getting the impression now from listening to all the comments-- from both the panel and the audience--that people seem to view this idea of global/local analysis as an interim measure pending the full-scale utilization of parallel processing--that as soon as computers become fast enough, there won't be a need to use global functions and exact solutions. We'll just go in there with brute force with 3-D finite element analysis. I'm wondering if that's the case, because if that is the feeling then perhaps we should retrain in midstream here. Maybe that's a rhetorical question. On the other hand, if we do continue with these global/local methodologies, I would like to see them become more problem oriented, I'd like to see them continue in this manner, I'd like these benchmark problems--the NASA problems--to be distributed among people that are collaborating in this area. And I would like to then compare the methods used to solve these problems based on a number of criteria, such as (1) reliability; (2) computational effort, which would include the analyst's effort as well; (3) implementational effort, that's important because retreading software can be a huge cost and, finally, (4) generality. If worse comes to worst, we will have to build special purpose codes as they do in the fluids business. But, optimally, it would be nice to hang on to the general purpose benefits we get from the finite element method by using such techniques as reduced basis global functions, based on finite element basis functions, in the first place, so that we can automate this global/localization.

Ahmed K. Noor, George Washington University: Just two comments. One with regard to the question about uncertainty and confidence in the results. Short of doing a stochastic analysis I think we can get some confidence in the results by carrying out a sensitivity analysis. Sensitivity of the response with respect to the input parameters, for example. This would give us some confidence in the numerical results. The second comment deals with parallel processing. Parallel processing is a system issue. Unfortunately, much of the work that is being done has addressed certain aspects of the analysis process. In

particular, the numerical algorithms for handling the algebraic equations. This is very important, of course, and computer time spent in algorithms is dominant in present-day finite element analysis, but it might not be enough to address that alone if we want a very efficient solution procedure. I think we have to address the question from a higher level starting with the formulation, looking at the different phases of the analyses, and, as I said in my presentation, that we might have to move away from some of the traditional finite element analysis processes like, for example, bypassing the assembly process. This is the most difficult part of the solution process for vectorization or parallelization. So, I think we have to look more into the overall computational strategy rather than just the numerical algorithm.

Nelson: This comment has to do with something we haven't talked a lot about, although we saw a lot of pictures. That was the idea of testing. I'm reminded of an old Tinius Olsen testing motto which says that one test is worth a thousand expert opinions. We want finite element analysis to be a true engineering tool. Which means that the finite element software has to be configured to be flexible. If, for example, we start looking at such things as a NASA test panel, in which we begin to see failure such as delamination and things of this nature, we want to be able to incorporate in the software new material models or new failure laws for composites. So then we can have tight feedback between experiment and analysis. We will all benefit from having codes that are not just bigger--the codes are just a little bit smarter, because we are.

J. N. Reddy, Virginia Polytechnic Institute: I want to make one comment in response to the first comment made by Dr. Backman. Most of my colleagues from universities, I hope, share at least the spirit of what I'm going to say. I think most of the students that are coming out of the universities, I am disappointed to say, are taught the recipe-type approach, formula plugging and checking compared to my older colleagues who have a lot of ingenuity and physics background. My older colleagues are better at modeling than, I think, the current students. And I don't want to use the same kind of approach as proposed or suggested by Dr. Backman. I want the students to know more physics than number crunching and formula plugging. I think if they have an understanding of the basic physics and if they have certain tools, when they get out of school I think they can be trained on the job. I'll give an example of my own. After I graduated, I took a job with Lockheed Huntsville Space Company. Before taking that job, I only worked with Hilbert spaces and Sobolev spaces, and also, of

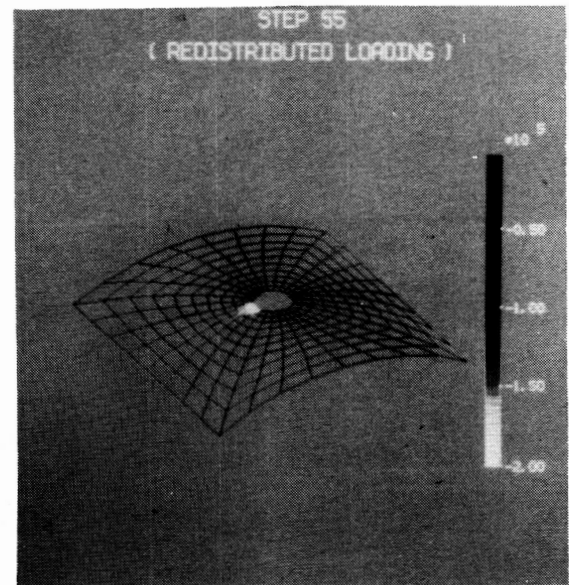
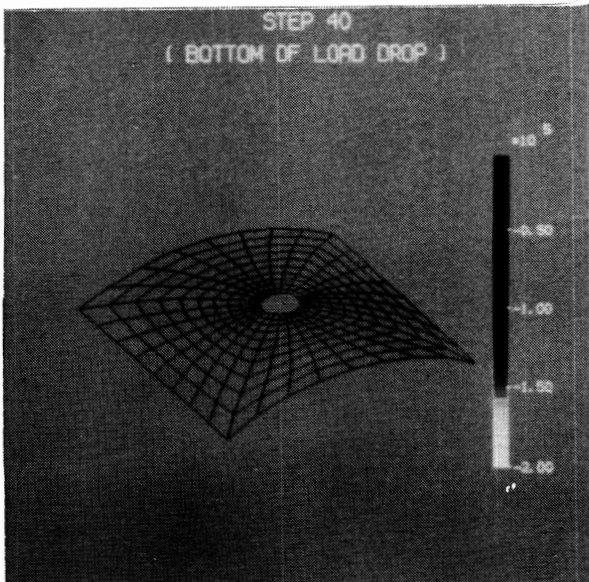
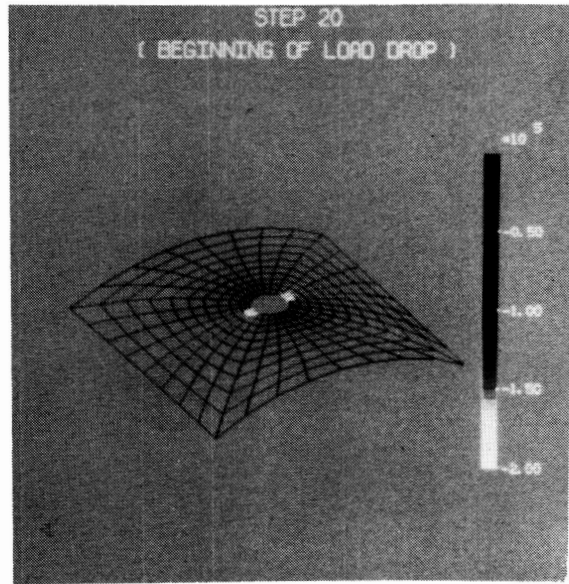
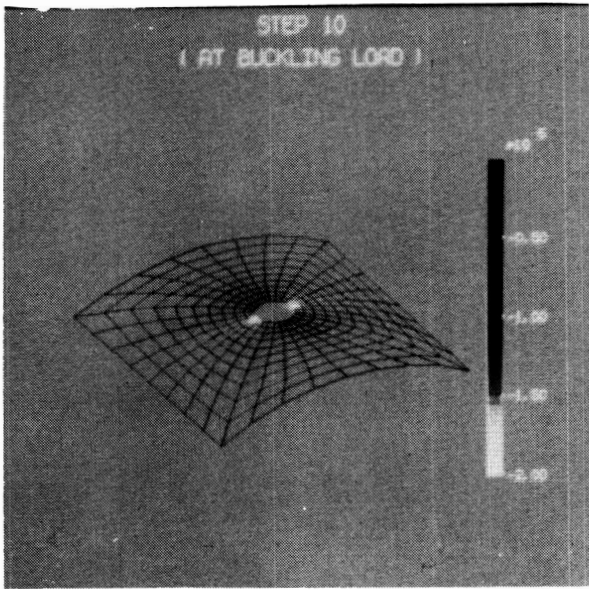
course, I learned a great deal from Professor Oden and other people in terms of physics and mathematical modeling. At my new job, I was asked to do a 3-D analysis of impact, hydrodynamic approximation. One week after I got into the job, I had a job offer to go to the University of Oklahoma, but I decided to stay and complete the project given to me. And I did complete it in 6 months. The point is, if you have the right background in terms of the mathematics and physics, I think you can learn number crunching very easily. But if you train yourself with only number crunching, you may not get the physical understanding. So I think we should be very careful in training our students.

Stroud: My bet is that Bjorn wasn't suggesting that students be trained a particular way. Rather, he wanted the product of our CSM efforts to be software.

Backman: Yes. I'm not sure what I said that caused this comment. It's possible that I hinted that the level of so-called blackbox engineering has to be increased. I think that we are slowly reaching the point where proliferation of experts has reached a level such that we simply can't replace them fast enough in the production field. And what I'm trying to say is, of all these students that are coming up with the proper background in science and structural mechanics and so on, how many do you expect to be proficient in the operating systems that exist on the machines they are using? How many do you expect to know the details of the compilers we use on the system? How many of them do you expect to understand, in detail, the equation solvers? And how many do you expect to understand the inner workings of the optimizers? Somewhere in here we are going to have to stop asking these people to become renaissance personalities. I'm just trying to achieve some balance here where maybe things like finite element systems ultimately fit into the same category as operating systems. And maybe the engineering community will be dealing with the actual design problems. I'm, by all means, in favor of a solid background in the sciences. I'd rather see the industry taking on some additional educational responsibility in this area. The point is that sooner or later you either have to require 10 years or more of higher education or you have to identify how deeply into the problem you want to go. And, I think, a method like this one that Dr. Szabo has been advocating is probably a first step towards the ability to heuristically understand the result of your analysis without knowing the guts of each step of the solution. And I'll stop with that.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH

Stroud: Thank you. It's clear that we have opened up a lot of topics and we have not resolved them. And opening them up is certainly one of the objectives of this workshop. There will be many more workshops and conferences that will attempt to resolve some of these questions. I want to thank you all for coming and participating. In particular, I want to thank the speakers.



*Original color slide shown here in black and white.

TIRE MODELING

PRECEDING PAGE BLANK NOT FILMED

**FEATURES AND CHARACTERIZATION NEEDS OF
RUBBER COMPOSITE STRUCTURES**

Farhad Tabaddor
Goodrich Company
Akron, Ohio

Abstract

This paper outlines some of the major unique features of rubber composite structures. The features covered are those related to the material properties, but the analytical features are also briefly discussed. It is essential to recognize these features at the planning stage of any long-range analytical, experimental, or application program.

The development of a general and comprehensive program which fully accounts for all the important characteristics of tires, under all the relevant modes of operation, may present a prohibitively expensive and impractical task at the near future. There is therefore a need to develop application methodologies which can utilize the less general models, beyond their theoretical limitations and yet with reasonable reliability, by proper mix of analytical, experimental, and testing activities.

OUTLINE:

1. CORDS
2. RUBBER
3. RUBBER COMPOSITES
Single Ply
Laminate
4. ANALYTICAL AND
COMPUTATIONAL ASPECTS

INTRODUCTION

The textile-cord-reinforced-rubber composites used in various industrial products differ in many respects from the classical rigid composites. There are a great number of papers, books, and professional journals devoted to rigid composites. The literature on rubber composites, however, is very limited as compared to classical composites. To understand the mechanics of such composites, it is essential to develop an in-depth understanding of the way in which the internal variables of the constituents participate and interact in responding to external agents, i.e., mechanical, thermal or other environmental forces. It is, however, important to first study the properties of each constituent before dealing with the material properties of rubber composites.

CORDS

The textile cord reinforcements are structural members which may be viewed as one dimensional in a microscopic sense. The cords generally consist of several yarns twisted together. The yarns also consist of numerous filament components organized in a geometrical array with the view towards enhancing certain target properties. The cord properties therefore depend on the properties of the filaments and the geometrical organization as well as the interfacial characteristics. The filament itself has also been known to possess an internal structure at the microscopic level.

In view of the above consideration, the cord itself is a complex structure which may be studied at the microscopic level (figs. 1 and 2). The way in which the cord properties are related to filament properties, geometries (ref. 1) and other variables is the subject of textile mechanics (fig. 3). Properties such as strength, stiffness, and fatigue characteristics can be controlled by internal variables when their relationships are well understood.

From a higher scale of continuum mechanics we are, however, concerned only with the phenomenological properties of the cords as experimentally obtained without any attempt for relating such

measured values to the micro-structure in the sense of textile mechanics. The properties of interest at this continuum level are therefore the effective cord properties.

Such an approach enables us to bypass the complexity of the textile mechanics in our formulation of composite properties. The limitation is, however, that such measured properties serve as "averages" only and therefore the continuum elements should be at least in the order of cord diameters.

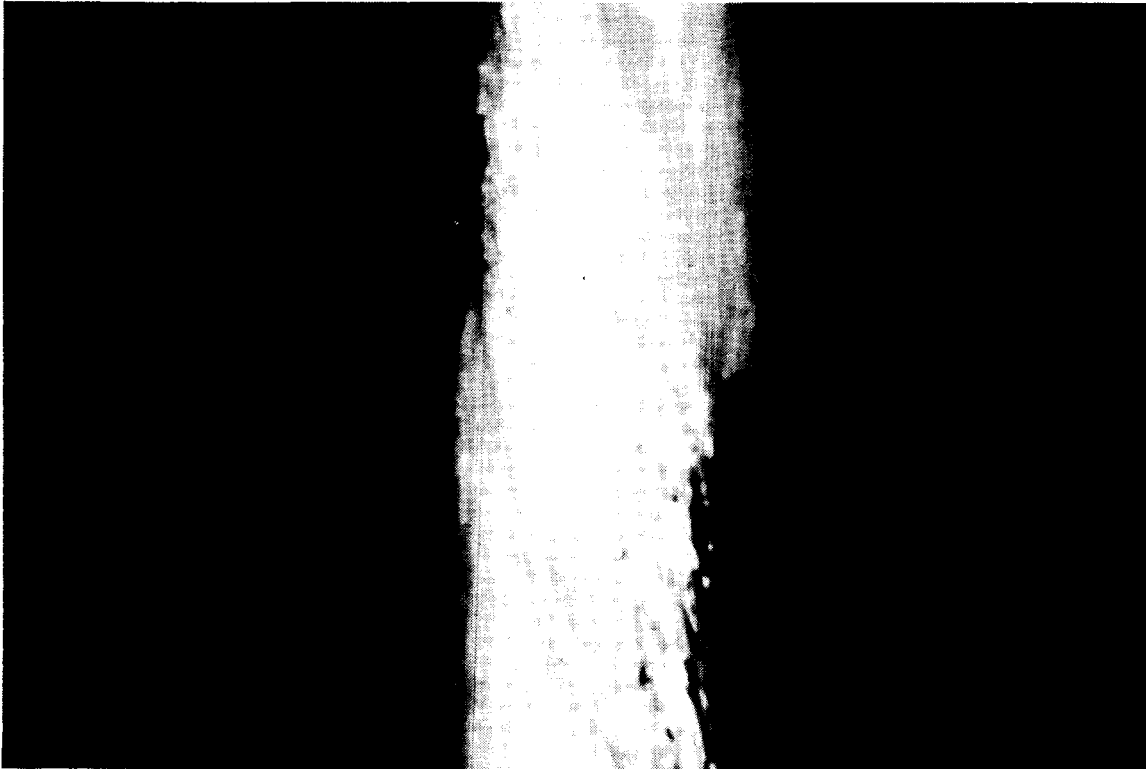
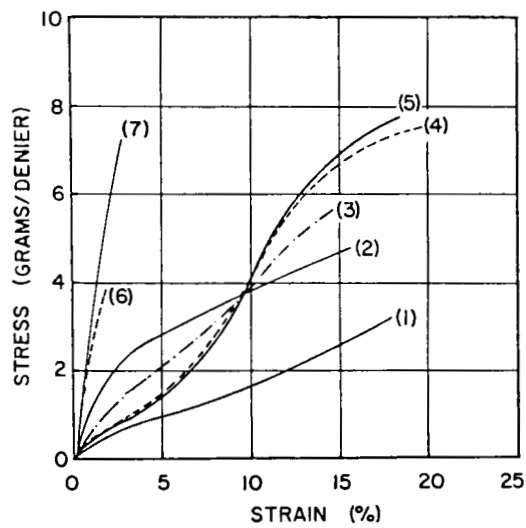


Figure 1. Typical tire cord.

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



Figure 2. Cross section of tire cord (2).



- | | |
|---------------|--------------|
| (1) RAYON | (5) NYLON-66 |
| (2) NOMEX | (6) STEEL |
| (3) POLYESTER | (7) GLASS |
| (4) NYLON-6 | |

TESTED AT 70°F, 65% R.H.
INSTRON - 100% STRAIN RATE

Figure 3. Stress-strain characteristics of typical tire cords (2).

The cord is the major load-bearing member of rubber composite structures and as such should provide strength and many other characteristics of interest. Some of the expected performance characteristics of the tire, and for that matter any other structure, can be directly related to cord properties. Let us consider some of the tire performance characteristics which are affected by cord properties. A partial list is given in Table 1.

Table 1

TIRE PERFORMANCE AFFECTED BY CORDS

- o Burst Strength
- o Bruise Resistance
- o Tire Endurance (Separations)
- o Power Loss
- o Tread Wear
- o High-Speed Endurance
- o Tire Size and Shape
- o Groove Cracking
- o Flat Spotting and Non-Uniformity
- o Tire Cornering Force
- o Tire Spring Rate
- o Noise

Tables 2 to 4 of reference (3) provide a list of tire requirements and the related cord requirements. Such relations should be viewed with caution and qualifications.

Table 2

(REF. 3)

RADIAL PASSENGER

Vehicle Trends	Tire Requirements	Cord Requirements
High Performance	Low Aspect Ratio (Increased Cornering Forces)	Modulus Lateral Stiffness
Downsizing	Downsizing - Monoply	Dimensional Stability Fatigue Modulus Tenacity
Front Wheel Drive	Improved Tread Wear Retreading	Dimensional Stability Lateral Stiffness Aged Adhesion Thermal Stability Fatigue
Fuel Economy	Rolling Resistance	Tenacity Modulus Hysteresis

Table 3

(REF. 3)

RADIAL TRUCK

Vehicle Trends	Tire Requirements	Cord Requirements
Increased Drive Position Loads	Increased Durability	Tenacity Strength/Area Ratio
Improved Fuel Economy/New Transportation Act	Lighter Weight Low Profile Retreading	Tenacity Fatigue Aged Adhesion Fatigue Thermal Stability
	Rolling Resistance	Tenacity Modulus Hysteresis

Table 4

(REF. 3)

RADIAL AIRCRAFT

Tire Requirements	Cord Requirements
High Deflection	Tenacity Modulus Thermal Stability
Reduced Weight	Tenacity Strength/Area Ratio
Retread	Aged Adhesion Dimensional/Thermal Stability Fatigue

CORD VISCOELASTICITY

The textile cords used in rubber-reinforced composites are often nonlinear viscoelastic. The viscoelastic deformations are associated with the loss of energy. The dissipated energy appears as heat and leads to temperature rise which in turn affects the material properties. The cord is therefore an important contributor to the energy loss in rubber-composite structures such as tires (4). Figures 4 and 5 show some of the viscoelastic properties of polyester cords.

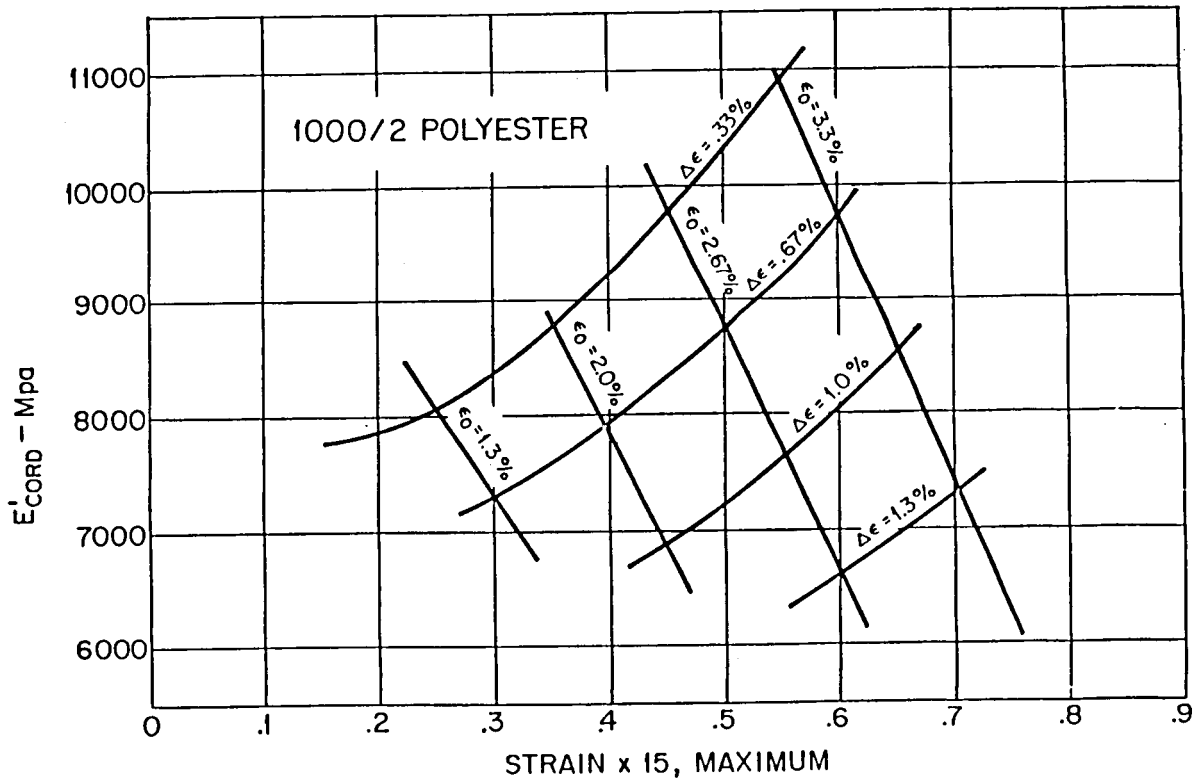


Figure 4. Carpet plot of 1000/2 polyester real modulus.

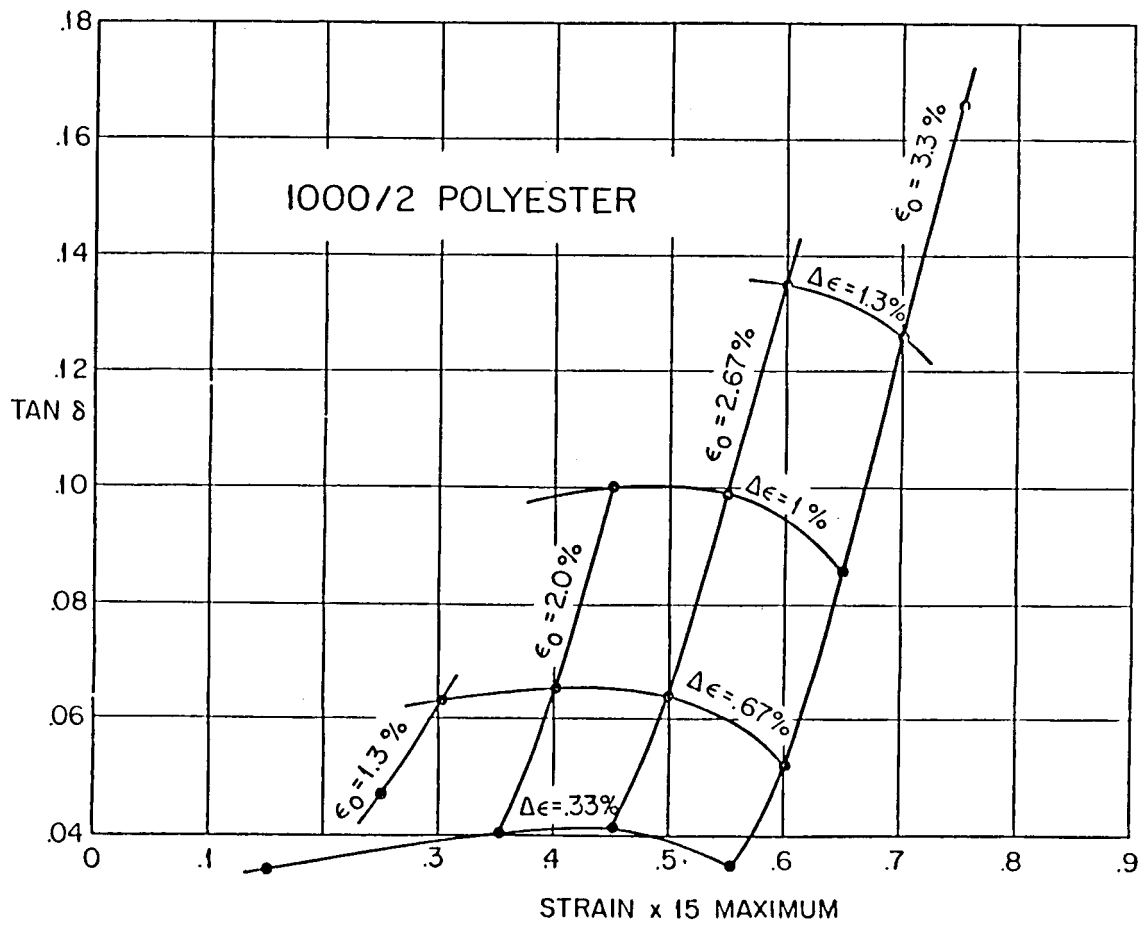


Figure 5. Carpet plot of 1000/2 polyester loss modulus.

SUMMARY ON CORDS DISCUSSION

- o Cords are structures and as such their effective properties are geometry and boundary condition dependent.
- o The effective cord properties are different when cords are embedded in the rubber matrix.
- o The calculation of the three-dimensional effective properties by analytical homogenization is very complex and impractical.
- o The properties of some cords such as nylon are highly temperature sensitive, particularly around the glass transition temperature.
- o The properties are different in tension and compression.
- o Cords are often nonlinear and viscoelastic.

RUBBER

The physical properties of rubber compounds depend on various processing parameters and components. Here again, we are not concerned as how these properties are related to the molecular structure of the rubber and the physics of rubber vulcanization. The focus is on determination of the relevant properties by proper experiment and within the framework of continuum physics. The most important property is the elasticity of the rubber, which is distinctly different from other conventional materials. The most distinctive features of rubber elasticity are the deformability and the rapid recovery of the deformations when loads are removed. Rubber remains elastic at extension ratios of several hundred percent. Such elastic characteristics make the rubber unique in this respect. In fact, the major developments in the theory of large elastic deformations evolved around application to rubber elasticity. The rubber elasticity is an important subject in understanding finite-element analysis of such products.

RUBBER ELASTICITY

To understand rubber elasticity, we may first examine the thermodynamics of reversible processes. The first and the second laws of thermodynamics state that (5)

$$dE = Tds + dw$$

where E is the internal energy, T is the absolute temperature, S is the entropy and w is the work done on the system. Experimental work has shown that the rubber elasticity resides basically in the entropy term. The rubber elasticity therefore has an entirely different molecular origin than other elastic materials whose elasticities are primarily associated with the increase in internal energy through changes in molecular or atomic spaces.

Much work has been carried out to formulate rubber elasticity. One example of the molecular approach is one which considers the molecular chain length having Gaussian distribution. The elasticity parameters are calculated from such quantities as finite molecular length and molecular weight between crosslinks. The entropy change resulting from Gaussian theory leads to

$$\Delta S = -\frac{1}{2}NK (\lambda_1^2 + \lambda_2^2 + \lambda_3^2 - 1)$$

in which N is the number of network chains per unit volume and λ_1 , λ_2 , and λ_3 are the three principal extension ratios along the three mutually perpendicular axes of strain for pure homogeneous strains.

The above equation provides a first order approximation to rubber elasticity but is not adequate over a large range of deformations.

PHENOMENOLOGICAL APPROACH

An entirely different approach (6) is the phenomenological approach of continuum mechanics. In this approach, the existence of a strain energy function is postulated. It has been shown that such a strain energy function should depend on deformation gradients. The equations for isotropic incompressible elastic media are as follows:

$$W = W (I_1, I_2, I_3)$$

$$I_1 = \lambda_1^2 + \lambda_2^2 + \lambda_3^2$$

$$I_2 = \lambda_1^2 \lambda_2^2 + \lambda_1^2 \lambda_3^2 + \lambda_2^2 \lambda_3^2$$

$$I_3 = \lambda_1^2 \lambda_2^2 \lambda_3^2$$

For the incompressible case

$$W = W (I_1, I_2)$$

$$I_3 = 1$$

In series form

$$W = \sum_{ij} C_{ij} (I_1 - 3)^i (I_2 - 3)^j \quad i, j = 0, 1, 2, \dots$$

An alternative form, suggested in (7), is as follows

$$W = \bar{W}(\lambda_1) + \bar{W}(\lambda_2) + \bar{W}(\lambda_3)$$

The stress-strain relations in the principal directions (the stresses are force per deformed area) are

$$\sigma_1 = \frac{\partial W}{\partial I_1} (2\lambda_1^2)^2 + \frac{\partial W}{\partial I_2} 2\lambda_1^2 (\lambda_1^2 + \lambda_3^2) + h(O)$$

$$\sigma_2 = \frac{\partial W}{\partial I_1} (2\lambda_2^2) + \frac{\partial W}{\partial I_2} 2\lambda_2^2 (\lambda_1^2 + \lambda_3^2) + h(O)$$

$$\sigma_3 = \frac{\partial W}{\partial I_1} (2\lambda_3^2) + \frac{\partial W}{\partial I_2} 2\lambda_3^2 (\lambda_1^2 + \lambda_2^2) + h(O)$$

The one-dimensional stress-strain relation can be obtained from the three-dimensional relations as shown in figure 6.

$$\sigma = 2 (\lambda^2 - 1/\lambda) \left(\frac{\partial W}{\partial I_1} + \frac{1}{\lambda^2} \frac{\partial W}{\partial I_2} \right)$$

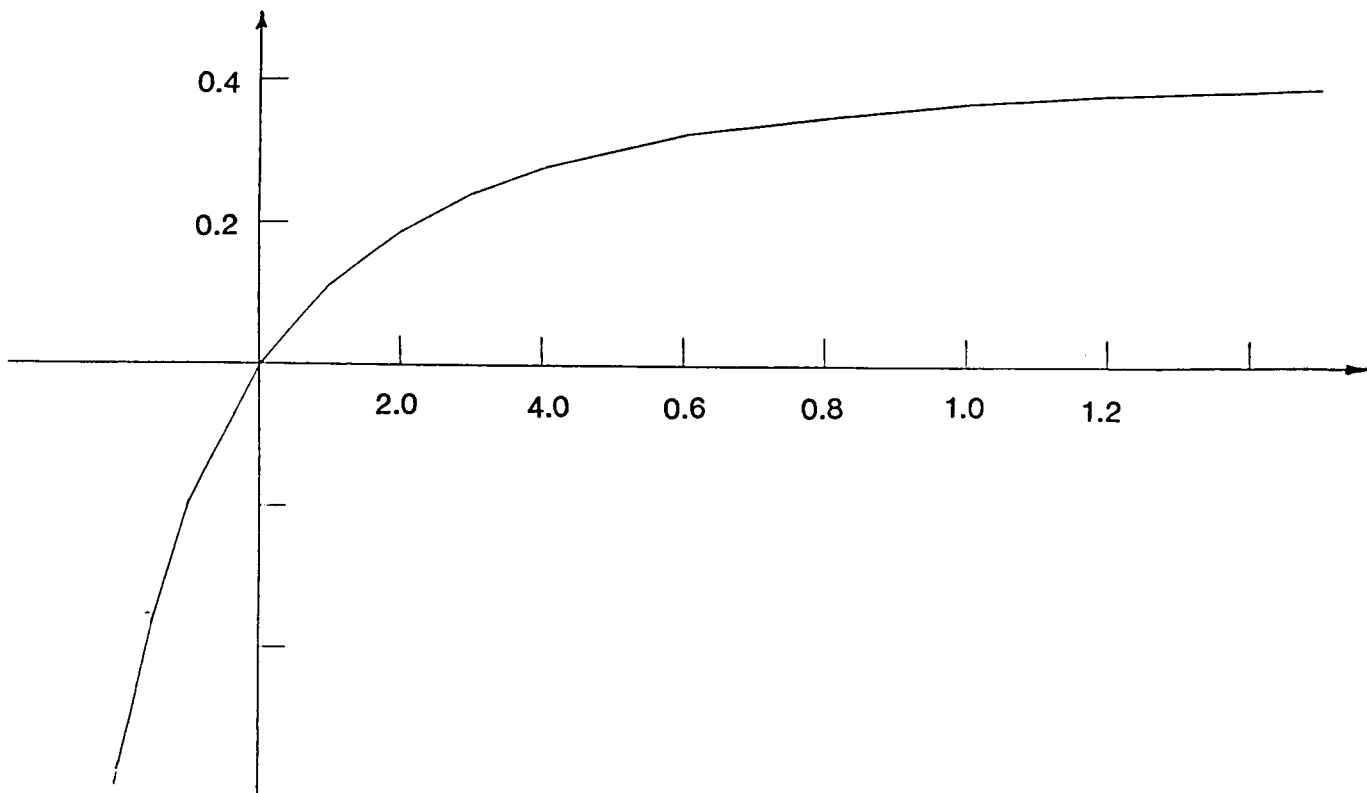


Figure 6. One-dimensional stress-strain relation.

It can be seen that the tangent compressive moduli increase significantly as the compressive strain increases. This feature realistically describes the rubber response in compression. In the finite-element treatment of rubber materials, the rubber is often modeled as linear, on the ground that the strains are small and that the linear constitutive law should then provide a reasonable approximation. This view is very disputable. It suffices to say that the compressive strains remain small due to stiffening of rubber in compression. This feature can only be handled properly by nonlinear constitutive laws. In the linear model of rubber, the material stiffness remains unchanged and therefore moderate compressive forces produce high compressive strains that are not seen in the actual structure. Such exceedingly large strains result in distortion of elements which quickly leads to severe numerical problems.

A difficult problem, in dealing with rubber elasticity, arises from the incompressibility constraint condition. The incompressibility condition leads to certain simplifications in the exact analysis of the problem, basically because of reduction in the number of unknown parameters. Such is not, however, the case with the finite-element approach. In the variational formulation, for example, the Lagrange multiplier introduces an additional unknown scalar function into the finite-element formulation. This unknown should be accommodated at the element level. The procedure results in a significant increase in the number of total unknowns and also may result in an ill-conditioned stiffness matrix. A great deal of research has been carried out to find the most suitable element for handling incompressibility. The subject is still open but the fact remains that the incompressibility imposes an additional burden in numerical analysis. The incompressibility condition, aside from being inconvenient in the finite-element analysis, is an approximation for rubber-like materials. Such an approximation becomes increasingly less accurate as the percentage of carbon black increases in the rubber compound. The exact enforcement of incompressibility is therefore not actually needed.

NEARLY INCOMPRESSIBLE MATERIALS

The strain energy function for nearly incompressible materials can be obtained by a series expansion of the strain energy function about $(I_3 - 1)$, and retaining the leading terms of up to second order in $(I_3 - 1)$, as follows

$$W = W_1 (I_1, I_2) + W_2 (I_1, I_2) \cdot (I_3 - 1) \\ + W_3 (I_1, I_2) \cdot (I_3 - 1)^2 + \dots$$

$$|I_3 - 1| \ll 1$$

$$W = \sum_{ij} C_{ij} (I_1 - 3)^i (I_2 - 3)^j + W_2 - (I_3 - 1) + W_3 (I_3 - 1)^2 + \dots$$

One special case of the above equations is when the function W_2 and W_3 are considered constants. The constants are described as follows:

$$W_2 (I_1, I_2) = H_1$$

$$W_3 (I_1, I_2) = H_2$$

$$H_1 = -(C_{10} + 2C_{01})$$

The near incompressibility can now be enforced by assigning large values to H . In fact, as H approaches infinity, (I_{3-1}) approaches zero so that the strain energy remains finite. The higher the value of H , the closer the incompressibility would be satisfied. H is referred to as the penalty number. In finite-element analysis, however, the large values of H can lead to overriding stiffness which results in numerical problems. The penalty method nevertheless permits the satisfaction of near incompressibility without increasing the number of unknowns in finite-element formulation. Figure 7 shows a set of typical properties for the rubber.

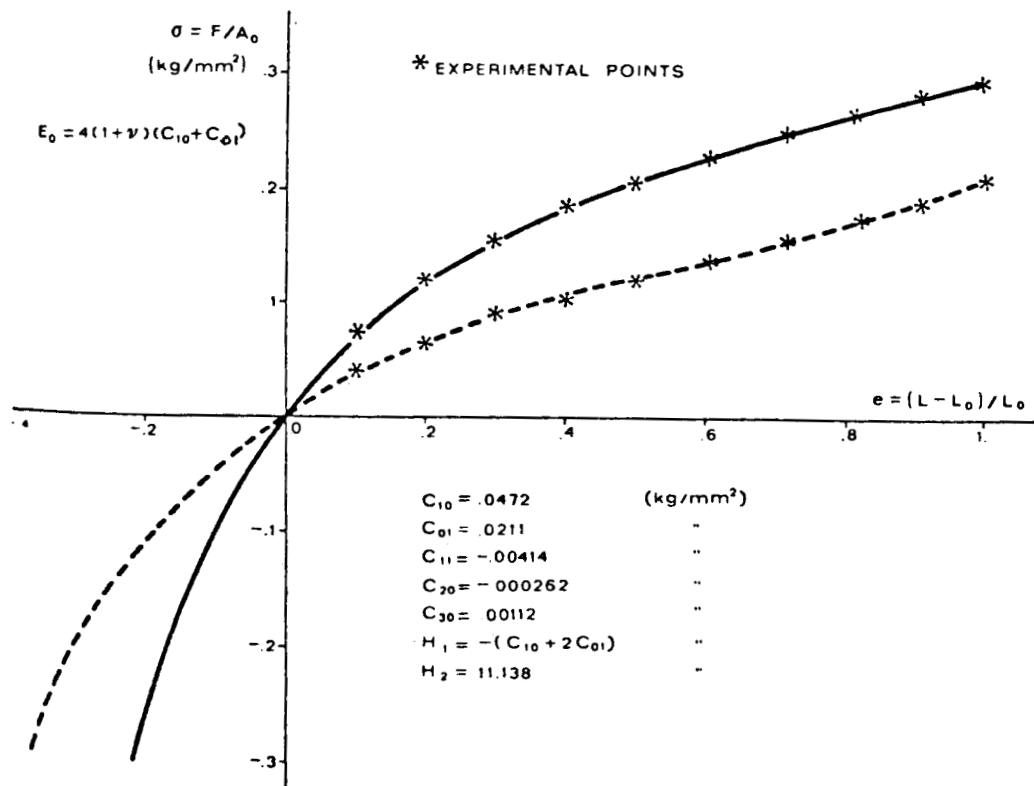


Figure 7. Stress-strain relation, (8).

FRACTURE AND FATIGUE PROPERTIES

The fatigue of the rubber has been the subject of many investigations in the past. The rubber fatigue is intimately related to the nature of rubber fracture and cut growth. The fracture mechanics approach for rubber was first adopted by Rivlin and Thomas (9) and Thomas (10) who promoted the concept of the tear energy in describing the cut growth mechanism. The tear energy approach has been applied to the study of the crack growth problem and to the description of fatigue behavior of rubber. We only consider the mechanically induced fatigue and this therefore excludes the fatigue caused by or resulting from non-mechanical sources such as aggressive environment. Some of the non-mechanical sources, such as ozone cracking in elastomers, may, however, be more damaging than mechanical sources.

Busse's early results (11) on NR compounds, shown in Table 5, clearly demonstrate the unique feature of rubber fatigue.

Table 5

EFFECT OF STRAIN CYCLE

STRAIN CYCLE	LIFE, MINUTE
5 - 60% Extension	10
10 - 60% "	25
15 - 60% "	90
17 - 60% "	150+
20 - 60% 00	00

Some experimental fatigue data on various compounds are shown in Figures 8 to 11 taken from References (12-14).

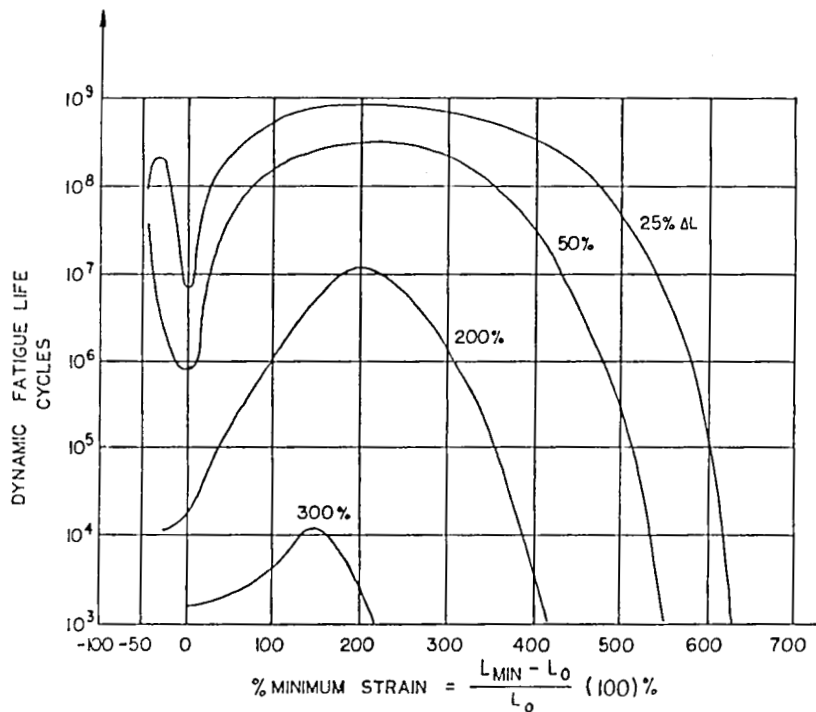


Figure 8. Fatigue life of NR as a function of minimum strain range (from ref. 12).

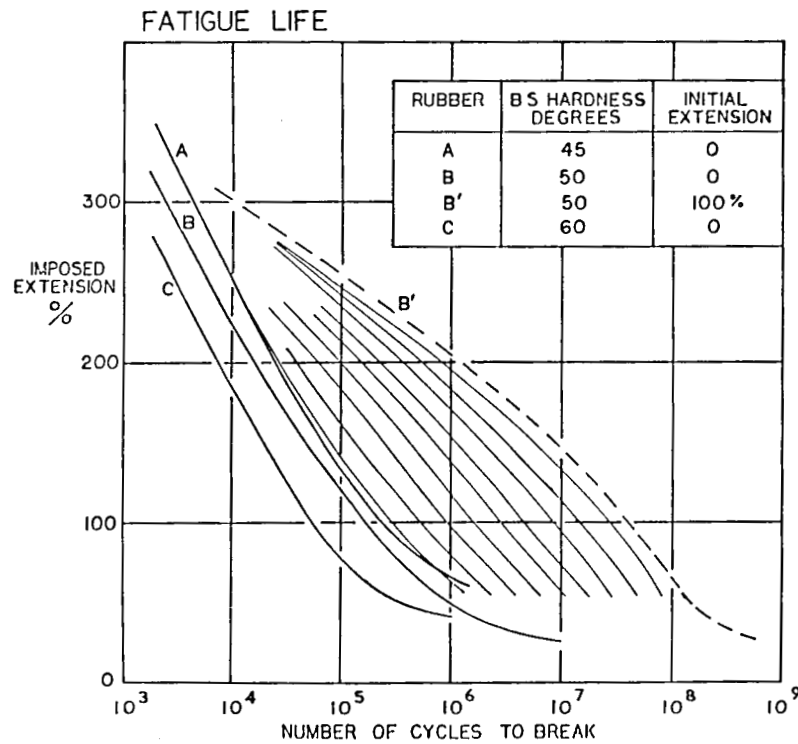


Figure 9. Imposed extension versus number of cycles to break (from ref. 12).

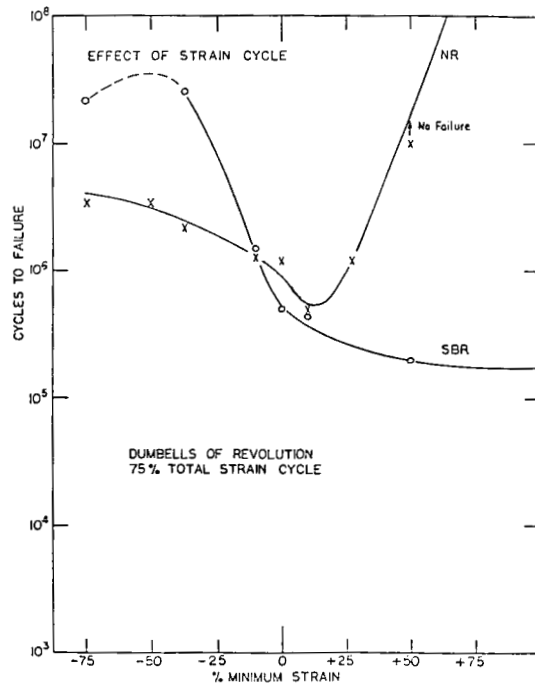


Figure 10. Cycle of failure versus minimum strain for NR and SBR (from refs. 13 and 14).

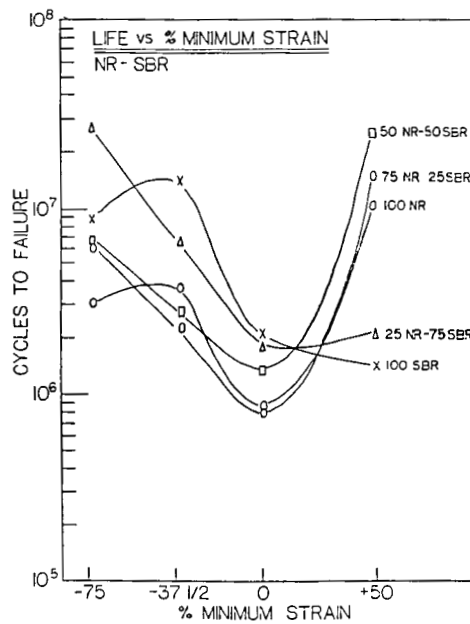


Figure 11. Cycle of failure versus minimum strain for various compositions (from refs. 13 and 14).

SUMMARY ON RUBBER

- o Rubber remains elastic at extension ratios of several hundred.
- o Rubber is almost incompressible.
- o The rubber elasticity can be most conveniently expressed through the strain energy function.
- o Rubber has unique fatigue properties. Extensive experimental work needs to be carried out for fatigue characterization of various rubber compounds.
- o Appropriate failure theories are needed for interpretation of stress data from finite-element analysis.
- o Properties are temperature dependent.

CORD REINFORCED RUBBER COMPOSITE - SINGLE PLY

In the macro-mechanics approach to composites, the actual heterogeneous medium, Figure 12, is replaced by an "equivalent" homogeneous medium. The equivalent medium has the same geometry as the heterogeneous medium but different material properties. The "effective" properties of the latter may belong to a different class of symmetry than those of the constituent materials. For instance, for isotropic constituents, the effective properties may be isotropic, orthotropic or anisotropic depending on the internal geometry of the composite.

Let us consider a two-dimensional composite made of an isotropic matrix and reinforced by doubly periodic arrays of isotropic fibers as shown in Figure 13. A composite representation of the problem is a homogeneous medium with the same external geometry and boundary conditions as those of the heterogeneous case, but with orthotropic properties. The effective orthotropic properties are then defined for a "representative" element. It is very important to first understand what is meant by that. Hashin (15) defined and discussed this question and has drawn some analogies to the case of homogeneous continua. In such continua, the hypothesis is that the continua retain their properties even for infinitesimal elements. Due to the discrete nature and microstructure of homogeneous media, the infinitesimal element of continuum mechanics should be large compared to the scales of the microstructure. The infinitesimal element therefore exhibits properties which are some statistical averages of the microstructure properties. On the other hand, the infinitesimal elements should be very small compared to the dimensions of the continua. It then follows that the physical quantities at a point, such as moduli, stress and strain components, are in reality associated with averages over infinitesimal elements and not with a geometrical point. Due to the complex nature of the microstructure, the properties of the continuum infinitesimal elements are determined experimentally as opposed to calculating the averages from a microstructural theory. The same comments also apply to the "representative" element of composite theories, that is to say that a "representative" element is the infinitesimal element of composite materials. It therefore should be large compared to the dimensions of its material phases. A "representative" element, as defined, would retain and represent the properties of the composite continua and furthermore these properties would be insensitive to boundary conditions.

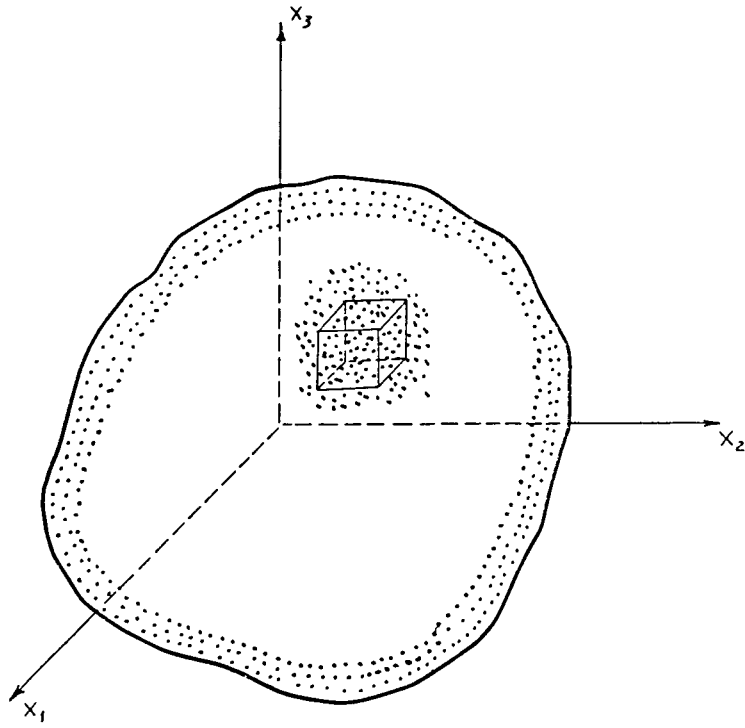


Figure 12. Representative element.

A distinction must be made between a "representative" element and unit elements or unit cells. The latter is defined as building blocks, so that the continua can be constructed by repeated use of such units. For example, consider the composite of Figure 13. There are a number of unit cell configurations which can equally serve as building blocks. Some of these possible choices are shown in Figure 14. Figure 15 shows the boundary effects on shear deformations. The average properties of these units, unlike those of a "representative" element, are highly boundary condition dependent (16). A "representative" element, however, consists of a large number of unit cells.

From the definition of a "representative" element, it is apparent that the properties calculated or experimentally measured would be rather insensitive to boundary conditions. The problem, however, is that such a calculation would be a formidable task. Most published works dealing with a calculation of effective properties use a unit cell as the basis of their computations. Even for unit cells, the exact solutions can be obtained for only some simple geometries and simple boundary conditions. Another approach adopted is to determine the upper and the lower bounds for these properties through approximate solutions of energy formulations. These bounds are, however, far apart for composites with high cord-to-matrix stiffness ratios, such as rubber composites, and are therefore of little practical use.

The immediate question is, therefore, how sensitive these properties are to boundary conditions specified over the surface of the unit cell, and, furthermore, how the element boundary conditions are influenced by global boundary conditions. These questions have not been fully investigated in the literature. It can, however, be stated that such sensitivities should depend on the relative stiffnesses of the cord and the matrix. As the stiffness of the matrix approaches that of the reinforcing materials, the effective properties should become less sensitive to boundary conditions and obviously independent of the boundary conditions in the limiting case of identical constituents' properties. These sensitivities are thus of greater concern in rubber composites than conventional rigid composites. The ratio of cord to rubber stiffness can exceed 30,000 in some composites which is far greater than those of rigid composites.

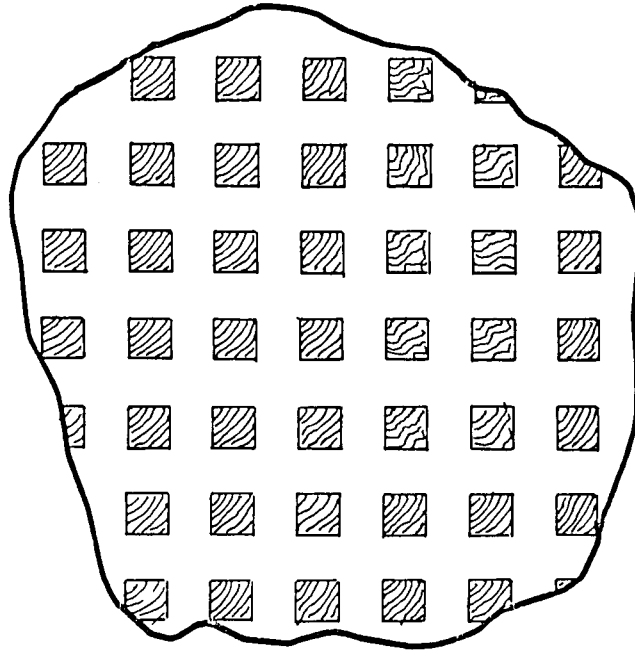
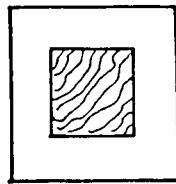
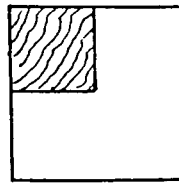


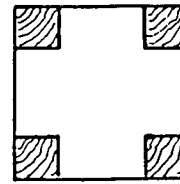
Figure 13. Composite with doubly periodic fiber arrays.



(I)

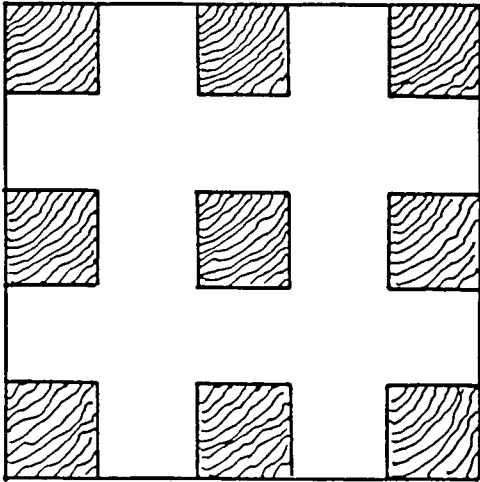


(II)

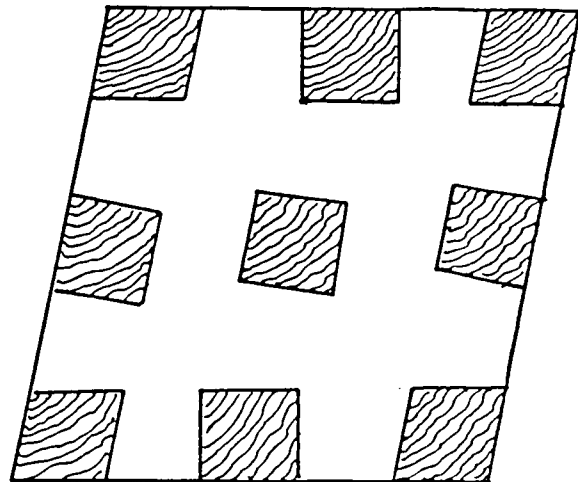


(III)

Figure 14. Several unit cells.



(a) Undeformed



(b) Deformed

Figure 15. Uniform shear strain boundary condition.

SUMMARY ON BASICS OF RUBBER COMPOSITES

- o The calculation of effective properties of rubber composites by homogenization is subject to more limitations than rigid composites. These limitations need to be established on a sound theoretical basis.
- o In experimental determination of the effective properties, the effects of size and the boundary conditions should be investigated.
- o The homogenized properties of a single ply cannot be easily calculated since all the 3-D homogenized cord properties are not generally available, with sufficient accuracy. Some of the effective composite properties, however, may prove to be somewhat insensitive to inaccuracies in cord properties.
- o A necessary characteristic of a composite material is statistical homogeneity. A representative element, used for calculation of the effective properties, should be large compared to typical phase regions. A representative element of a single ply does not have such a characteristic in the thickness direction. The out of plane properties of a single ply, therefore, are subject to question.
- o The effective properties are different in tension and compression.

CHARACTERIZATION OF LAMINATED COMPOSITES

We now consider a laminated structure, composed of N layers of cord-reinforced composite materials, as shown in **Figure 16**. Each layer of heterogeneous composite may then be modeled as homogeneous but orthotropic with respect to the proper local coordinate of each layer. The constitutive equations of the laminated composite, however, must be obtained with respect to a global coordinate system XYZ , as shown in **Figure 16**. The transformation relations may then be used for the appropriate layers to carry out the required transformation. The layers are numbered from top to bottom and no symmetry is assumed with respect to any axis. The stress and moment resultants for the laminated structure in terms of stresses are defined as follows:

$$\begin{aligned} N_x &= \int_0^t \sigma_{xx} dz & N_y &= \int_0^t \sigma_{yy} dz & N_{xy} &= \int_0^t \sigma_{xy} dz \\ M_x &= \int_0^t \sigma_{xx} z dz & M_y &= \int_0^t \sigma_{yy} z dz & M_{xy} &= \int_0^t \sigma_{xy} z dz \\ Q_x &= \int_0^t \sigma_{xz} dz & Q_y &= \int_0^t \sigma_{yz} dz \end{aligned}$$

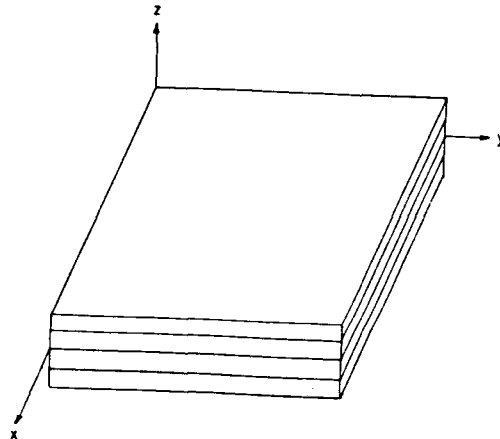


Figure 16. Geometry of typical laminated composites.

The displacement components and the resulting constitutive relations are

$$u = u_0(x, y) + zk_x(x, y)$$

$$v = v_0(x, y) + zk_y(x, y)$$

$$w = w(x, y)$$

$$\begin{bmatrix} N_x \\ N_y \\ Q_x \\ Q_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{bmatrix} = \begin{bmatrix} A_{11} & A_{12} & 0 & 0 & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & 0 & 0 & A_{26} & B_{12} & B_{22} & B_{26} \\ 0 & 0 & A_{44} & A_{45} & 0 & 0 & 0 & 0 \\ 0 & 0 & A_{54} & A_{55} & 0 & 0 & 0 & 0 \\ A_{16} & A_{26} & 0 & 0 & A_{66} & B_{16} & B_{26} & B_{66} \\ B_{11} & B_{12} & 0 & 0 & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & 0 & 0 & B_{26} & D_{21} & D_{22} & D_{26} \\ B_{16} & B_{26} & 0 & 0 & B_{66} & D_{16} & D_{26} & D_{66} \end{bmatrix} \begin{bmatrix} \frac{\partial u_0}{\partial x} \\ \frac{\partial v_0}{\partial y} \\ \frac{\partial w}{\partial y} + k_y \\ \frac{\partial w}{\partial x} + k_x \\ \frac{\partial u_0}{\partial x} + \frac{\partial v_0}{\partial y} \\ \frac{\partial k_x}{\partial x} \\ \frac{\partial k_y}{\partial y} \\ \frac{\partial k_y}{\partial x} + \frac{\partial k_x}{\partial y} \end{bmatrix}$$

where the laminate stiffnesses are related to the layer stiffness by:

$$A_{ij} = \int_0^t \sum_{k=1}^n C_{ij}^{(k)} dz \quad i, j = 1, 2, 6$$

$$B_{ij} = \int_0^t \sum_{k=1}^n C_{ij}^{(k)} z dz \quad i, j = 1, 2, 6$$

$$D_{ij} = \int_0^t \sum_{k=1}^n C_{ij}^{(k)} z^2 dz \quad i, j = 1, 2, 6$$

$$A_{ij} = \int_0^t \sum_{k=1}^n C_{ij}^{(k)} k_{ij} dz \quad i, j = 4, 5 \quad \text{No summation on } i \neq j$$

SUMMARY ON LAMINATED COMPOSITES

- o Each ply may be homogenized, subject to the same limitations as those of a single ply.
- o Complete homogenization is not possible since the coupling between the forces and the moments can only be accounted for by preserving nonhomogeneity in the thickness direction.
- o Classical kinematics constraints, such as the Kirchhoff hypothesis, do not apply.
- o Plate/shell type stiffnesses are extremely hard to measure.
- o Constitutive relations are nonlinear due to angle change between the cords of adjacent plies.

ANALYTICAL AND COMPUTATIONAL FEATURES

KINEMATICS

The rubber composite structures undergo large strains as well as large rotations. The major kinematic features are listed in table 6.

Table 6

KINEMATIC FEATURES

LARGE RUBBER STRAINS

LARGE ROTATIONS

CORD ANGLE CHANGES

NEAR INCOMPRESSIBILITY CONDITION

CONSTITUTIVE RELATIONS FOR COMPOSITES

Single ply

The single-ply composites can be considered as linear orthotropic when referred to axes of symmetry. For large displacements but moderately small strains, the rubber and the cords can be considered linear, but the strain-displacement gradient relations are nonlinear. The most appropriate form of constitutive equations, in such case, is

$${}^t S_{ij} = {}^t C_{ijkl} {}^t \epsilon_{kl}$$

where ${}^t C_{ijkl}$ are orthotropic properties referred to coordinates of initial material symmetry. The term ${}^t S_{ij}$ are the components of the second Piolla-Kirchhoff stress tensor and ${}^t \epsilon_{kl}$ are the components of the Lagrangian strain tensor. This form is invariant under a rigid body motion and therefore needs to be updated due to the cord angle change. This does not hold for constitutive equations

which utilize other stress and strain measures. The major drawback of modeling each ply separately is the increased size of the finite-element problem.

Several plies

It is often more convenient to lump several plies together in one element and therefore reduce the size of the problem. The preceding equation can be utilized but not longer remains unchanged due to the change in the angle between the minus and the plus ply. In such models the orthotropic properties of the combined plies continually change as functions of the cord angle.

Composites with nonlinear constituents

It is often necessary to account for the rubber nonlinearities in the finite-element analysis, even for small strains. The reason for such a need is that the typical compressive forces can produce large rubber strains if the rubber stiffening is not properly accounted for in the material modeling. These unrealistic large strains may lead to element distortion and eventual loss of numerical stability in the finite-element model. No rigorous nonlinear composite constitutive equations have appeared in the literature for rubber composites. This awaits further research in this field.

Various modeling levels may be required depending on the nature of the problem. These models and their respective features are listed in Table 7.

Table 7

MODELING FEATURES

<u>ELEMENT TYPE FOR COMPOSITES</u>	<u>FEATURES</u>
MEMBRANE ELEMENTS	REQUIRES 2-D MATERIAL PROPERTIES, INTERPLIES MODELED INDEPENDENTLY ONE ELEMENT PER PLY AND ONE ELEMENT PER INTERPLY IN THE THICKNESS DIRECTION, NO NEED FOR MATERIAL UPDATE DUE TO THE CORD ANGLE CHANGE
TWO- AND THREE-DIMENSIONAL ELEMENTS REPRESENTING:	
SINGLE PLY	REQUIRES 3-D MATERIAL PROPERTIES, ONE ELEMENT PER PLY IN THICKNESS DIRECTION, NO NEED FOR MATERIAL UPDATE DUE TO THE CORD ANGLE CHANGE
SEVERAL PLYS	REQUIRES 3-D MATERIAL PROPERTIES, LESS ELEMENTS, MATERIAL PROPERTIES MUST BE UPDATED DUE TO THE CORD ANGLE CHANGE, CANNOT MODEL THE COUPLING BETWEEN THE BENDING AND IN-PLANE FORCES
SHELL-TYPE ELEMENTS	INCREASED MATERIAL PROPERTY INPUT, LEAST NUMBER OF ELEMENTS, CAN MODEL THE COUPLING BETWEEN THE BENDING AND IN-PLANE FORCES, KINEMATICS CONSTRAINTS OF THE CORRESPONDING SHELL THEORY, MATERIAL PROPERTIES MUST BE UPDATED DUE TO THE CORD ANGLE CHANGE

SUMMARY OF MAJOR ANALYTICAL AND COMPUTATIONAL NEEDS AND FEATURES

- o Large elastic strains.
- o Large rotations.
- o Incompressibility condition, natural rubber can be considered as incompressible, but filled rubber exhibits compressibility.
- o Nonconservative loading.
- o Element type and aspect ratio.
- o Self-adoptive schemes for load increments and step size.
- o Contact algorithms for frictional loadings.
- o Finite-element formulation in rotating coordinate system.
- o Substructuring for localized analysis.
- o Stresses are very erratic at regions of sudden change in stiffness, such as cord-rubber interface. When calculated from the finite-element displacement method, proper smoothing algorithms must be developed for nonlinear problems.
- o Sensitivity analysis for uncertain input, material properties or other variables.

REFERENCES

1. Riding G. "Study of the Geometrical Structure of Multi-Ply Yarns," *Journal of Textile Institute*, vol. 52, pp. 366-381, 1961.
2. Clark, S.K., "Mechanics of Pneumatic Tires," National Bureau of Standards Monograph 122, First Edition, 1971.
3. Oblath, R.M., "Toward 21st Century - The Challenges Ahead of Tire Reinforcements," Akron Rubber Group, Akron, Ohio, Jan. 26, 1984.
4. Tabaddor, F. and Clark, S.K., "Linear and Nonlinear Viscoelastic Constitutive Relations of Cord Reinforced Elastomers," Proceedings of the International Symposium on Constitutive Equations for Non-Linear Materials, University of Arizona, Tuscon, Jan. 1983.
5. Treloar, L.R.G., "The Elasticity of a Network of Long Chain Molecules (I and II)," *Trans. Faraday Society*, vol. 40, p. 59, 1944.
6. Rivlin, R.S. and Saunders, D.W., "Large Elastic Deformations of Isotropic Materials - VII. Experiments on the Deformation of Rubber," *Phil. Trans. Royal Society*, vol. A243, p. 251, 1948.
7. Valanis, K.C. and Landel, R.F., "The Strain Energy Function of a Hyper-Elastic Material in Terms of the Extension Ratios," *Journal of Applied Physics*, vol. 38, no. 7, pp. 2997-3002, 1967.
8. Tavazza, G. and Rimondi, G., "Stress and Strain Calculations of Rubber Goods," Proceedings of the International Rubber Conference, Kiev USSR, 1978.
9. Rivlin, R.S. and Thomas, A.G., *J Polymer Science*, vol. 10, p. 291, 1953.
10. Thomas, A.G., *J. Polymer Science*, vol. 31, p. 467, 1958.
11. Busses, W.F., *Ind. Eng. Chem.*, vol. 26, p. 1194, 1934.
12. Cadwell, S.M., Merril, R.A., Sloman, C.M. and Yost, F.L., *Ind. Eng. Chem. Anal.*, vol. 12(1), p. 19, 1940.
13. Beatty, J.R., *Rubber Chemistry and Technology*, no. 5, pp. 1341-1346, Dec. 1964.
14. Beatty, J.R., *J. of Elastomers and Plastics*, vol. 11, p. 147, April 1974.
15. Hashin, Z., "Analysis of Composite Materials--A Survey," *Journal of Applied Mechanics*, vol. 50, September 1983.

16. Tabaddor, F., "Mechanical Properties of Cord-Rubber Composites," Composite Structures, vol. 3, pp. 33-53, 1985.

NONLINEAR HIERARCHICAL SUBSTRUCTURAL PARALLELISM
AND COMPUTER ARCHITECTURE

Joe Padovan
The University of Akron
Akron, Ohio

Abstract

This paper investigates computer architecture in conjunction with the algorithmic structures of nonlinear finite-element analysis. To help set the stage for this goal, the development is undertaken by considering the wide-ranging needs associated with the analysis of rolling tires which possess the full range of kinematic, material and boundary condition induced nonlinearity in addition to gross and local cord-matrix material properties.

1. Introduction

With the advent of the finite-element method (FEM), the analysis of large-scale structure is finally possible. While large-scale linear finite-element simulations are relatively economical, such is not the case for nonlinear situations involving geometric, material and boundary induced nonlinearity¹⁻⁴. There are numerous aerospace and commercial structures which require full-scale nonlinear analysis to enable their improved design. This includes such structural systems as gas turbines, space structures, aircraft structure, autos, etc. Perhaps the most commonplace of such structures is the tire, which serves as a component to a wide variety of aerospace and auto systems.

To bypass the difficulties associated with nonlinear FE analysis, significant work has been channeled into two main areas, namely:

- i) The development of algorithmic improvements, element-element,⁵ constrained Newton/Raphson (NR),⁶ and hierarchical least squares,⁷
- ii) The design of new computer architecture enabling hardware speedup, i.e., as in vector processors (Cray, Cyber 205 and true parallel machines^{8,9})

In the context of such thrusts, not enough effort has been undertaken to consider how algorithmic structures might effect machine architecture or vice versa.

Based on the foregoing comments, this paper will investigate machine architecture in conjunction with algorithmic structure. To achieve this goal, the development will be undertaken by considering the wide-ranging needs associated with the analysis of tires. This approach was taken since, as will be seen in later sections, the needs of tire modeling embody essentially all the requirements of nonlinear continuum mechanics, namely¹⁰

- i) Material nonlinearity
- ii) Inelastic behavior

- iii) Large deformation/strain kinematics
- iv) Complex inertial fields
 - v) Nonlinear boundary conditions
- vi) Microstructure
- vii) Thermomechanical response
- viii) Solid fluid interaction

All this leads to the development of what is called hierarchical substructural parallelism which enables bottom-up/top-down modeling.¹¹ Overall a nonlinear multilevel substructuring scheme is overviewed which enables the simplification of the data based management (DBM) of parallel-type operators while still yielding enhanced computational speeds as well as reducing core requirements.

In the sections that follow, detailed tire modeling discussions embody the diversity of needs of nonlinear simulations, various types of current machine architectures, and potentials of hierarchical substructural parallelism. Examples that define enhanced properties will also be given.

2. Shortcomings of FEM Vis-à-vis Tire Structural Analysis

Noting Figure 1, the tire possesses a very regionalized/substructural form of construction. Overall it consists of:¹²

- i) Carcass plies, steel/glass/Kevlar cord-rubber composites
- ii) Belt plies (same as above)
- iii) Bead, bundled steel cords
- iv) Thread configuration
 - v) Regionalized rubber types
- vi) Belt edges, turnup plies

The operating environment consists of:

- i) The tire-road interface which involves varying pavement textures, flexibilities and resulting frictional characteristics^{13,14}
- ii) The tire-rim interface
- iii) The tire-rim-suspension behavior
- iv) Cornering, braking and accelerating maneuvers
- v) Standing, steady/transient rolling^{10,13-15}
- vi) Obstacle/hole envelopment roll over events^{13,14}
- vii) Pressurization^{16,17}

As seen from Figures 2 and 3, the pressurization and subsequent loading into standing contact can lead to large deformations and associated rotations. For instance Table 1 illustrates comparisons of the deflection fields generated from linear and nonlinear FE simulations.

In this context, it follows that there are several sources of response nonlinearity, namely

- i) Large deformation kinematics
- ii) The road-tire-rim interfaces
- iii) Bimodular behavior of cord-rubber composites in transitions from tension to compression
- iv) Thermomechanical interactions
- v) Material nonlinearity
- vi) Local large strain levels in various regions of the tire; belt edges, bead region, and tread
- vii) Dynamic impact interactions

Each of the foregoing sources of nonlinearity initiates different forms of response behavior.

For instance, from a kinematics point of view, the pressurization process causes rotations and deflections which lead to an overall stiffening of the tire. Similarly, as with Hertzian contact problems, the tire-road interface also exhibits hardening-type properties, namely, the hub force-deflection response is stiffening in character as noted in Figure 4.

In addition to the foregoing modeling difficulties, in general the tire response needs to be handled in several levels, namely

- i) Cord-matrix and regionalized rubber interfaces
- ii) Whole cord-rubber plies/laminae
- iii) Full laminate structures, several plies as in belt and carcass laminates
- iv) Full (global) structure

As one proceeds from (i)-(iv), a "bottom-up" modeling¹¹ approach is required wherein fine detail is handled at the lowest level while the upper level models are increasingly coarser so as to reduce overall degrees of freedom in a global model. Once the global-level model is solved what is needed is a "top-down" scheme¹¹ to provide proper mechanics information at the constituent level. Such an approach is necessary if proper stress and strain fields are to be captured hence enabling proper description of internal fields.

Current FE models of tires start from level (iii) and proceed to (iv). In this way, a true local-level description of mechanical fields is not possible.

3. Types of Parallelism

Multiprocessor computers fall basically into two main categories, namely

- i) Vector processors (Cray, Cyber 205)
- ii) True parallel processors (Flex, Goodyear)

Compared with single processor units (IBM 3084, CDC7600), vector processors enable quicker more efficient handling of matrix manipulations. This is achieved through the use of multiple processors which operate simultaneously on a succession of matrix elements. Data transfer for such operations is typically from a single common core storage.

In true parallel processors, different functions/operations are performed in separate processors. In such machines data transfer usually involves both a common core as well as individual local processor cores. For such machines very high speeds can be realized.

In the context of programming languages, vector processors typically can be programmed in enhanced versions of FORTRAN or the like. For true parallel processors, overall programming is generally achieved at two levels. At the local processor level, languages such as FORTRAN can be employed. At the total system level, machine control language MCL is usually employed.

4. Classical Solution Algorithm

The solution of large-scale FE simulations typically involves either some variant of the Newton/Raphson scheme NR, or an explicit/implicit time integration procedure. For the current demonstration purposes, the presentation will concentrate on static equation solvers. The most recent improvements for such problems fall into several categories, namely

- i) Element-by-element preconditioners (Hughes et al.⁵)
- ii) Constrained NR procedures of Padovan and Arechaga⁶
- iii) Constrained hierarchical least-squares algorithms of Padovan and Lackney⁷

Assuming large deformation kinematics along with potential material nonlinearity, the governing FE formulation takes the form^{6,7}

$$\underline{\tilde{F}} = \underline{\tilde{G}} + \int_R [\underline{B}^*]^T \underline{\tilde{S}} dv \quad (1)$$

where $\underline{\tilde{S}}$ is the second Piola Kirchoff stress tensor, $\underline{\tilde{F}}$ is the nodal force vector and $\underline{\tilde{G}}$ is the vector of body forces. Typically (1) is nonlinear and must be solved via NR schemes. After expansion into truncated Taylor series, (1) yields the following NR algorithm namely^{1,2,6,7}

$$\Delta \underline{\tilde{G}} + [\underline{K}_i] \Delta \underline{Y}_{i+1} = \underline{\tilde{F}} + \Delta \underline{\tilde{F}} - \int_R [\underline{B}_i^*]^T \underline{\tilde{S}}_i dv \quad (2)$$

where $[\underline{K}]$ defines the tangent stiffness matrix, that is^{6,7}

$$[\underline{K}_i] = \int_R \{ [\underline{G}]^T [\underline{S}_i] [\underline{G}] + [\underline{B}_i^*]^T [\underline{D}_{Ti}] [\underline{B}_i^*] \} dv \quad (3)$$

such that $[\underline{S}_i]$ is the prestress matrix and $[\underline{D}_{Ti}]$ is the tangent material stiffness. As noted earlier, the solution involves either the use of constrained procedures⁶ for appropriate load increment control or a direct Gaussian-type inversion scheme.¹

To date such methodologies have been employed either in single processor or vector processor machines. The shortcomings of the FEM outlined in the previous section are essentially a direct outgrowth of the limitations of the architecture of single and vector-processor-type machines. In the next section, the intrinsic structure of the INR algorithm will be explored to define new computer architectures to bypass such difficulties.

5. Hierarchical Substructuring

From a conceptual point of view, the INR scheme defined by (2) does not confine the FEM scheme to a particular type of computer configuration. Rather the problems of speed and storage are essentially hardware based. Specifically the main questions and problems evolve out of the need to define architectures which enable the use of multiple processors so as to enhance overall machine speed as well as memory size. While the CRAY and CYBER systems are certainly a step in the right direction, they fall short of the ultimate requirements. Currently very large-scale FE models can easily outstrip the available core storage and machine CPU speeds.

In seeking to develop new computer architectures one is faced with the fact that

- i) Vector processors require extensive cores as well as complex logic flows
- ii) True parallel processors still await the fruition of properly organized DBM

Based on the foregoing, this paper seeks to develop what is called a hierarchical form of substructural parallelism. Following the pioneering efforts of the NASA Langley group^{8,9} specifically, a nonlinear FE simulation, say of the tire, can be logically divided into a hierarchy of substructural groups defined by a variety of attributes, namely

- i) Material group
- ii) Geometric configuration
- iii) Kinematic behavior
- iv) Boundary conditions

At the lowest rung of the hierarchy, items (i)-(iv) are employed to define the specific local level substructural groups. The choice of the number of first-order groups is contingent on:

- i) Minimizing core requirements of local level processors
- ii) Minimizing number of perimeter nodes so that higher order substructural groups also have reduced core requirements for associated processors.

As can be seen, the main thrust is to maintain in core solutions for each local substructural processor.

Noting Figure 5, a given FE simulation can be broken up into a number of substructural levels. At each level internal nodes are eliminated to enable assembly through perimeter nodes. In terms of (2), the NR algorithm and its constrained counterpart can be substructured to yield the following first-level algorithms, that is:

$$\Delta F_{i+1}^{(1,k)} = [K_i^{(1,k)}] \Delta Y_{i+1}^{(1,k)} + \Delta G_{i+1}^{(1,k)} \quad (4)$$

$k = 1, 2, \dots$ Number of first-level substructure such that

$$\Delta \tilde{F}_{i+1}^{(1,k)} = \tilde{F}_{i+1}^{(1,k)} - \int_R(1,k) [B_i^*]^T \tilde{S}_i dv \quad (5)$$

$$\Delta \tilde{G}_{i+1}^{(1,k)} = \int_R(1,k) [N]^T \Delta \tilde{F}_{i+1} dv \quad (6)$$

where $()^{(1,k)}$ denotes the first level k^{th} substructure, $()_{i+1}$ the $(i+1)^{\text{th}}$ iteration, $\Delta \tilde{F}_{i+1}^{(1,k)}$ the nodal load increment, $[K_i]$ the substructural tangent stiffness, $\Delta \tilde{Y}_{i+1}^{(1,k)}$ the nodal deflection increment and $\Delta \tilde{G}_{i+1}^{(1,k)}$ the body force increment.

To enable assembly into second-order substructural groups, (4) is partitioned into internal and perimeter nodes yielding

$$\Delta \tilde{F}_{i+1}^{(1,k)} \rightarrow (\Delta \tilde{F}_{Pi+1}^{(1,k)} \quad \Delta \tilde{F}_{Ii+1}^{(1,k)}) \quad (7)$$

$$\Delta \tilde{Y}_{i+1}^{(1,k)} \rightarrow (\Delta \tilde{Y}_{Pi+1}^{(1,k)} \quad \Delta \tilde{Y}_{Ii+1}^{(1,k)}) \quad (8)$$

$$\Delta \tilde{G}_{i+1}^{(1,k)} \rightarrow (\Delta \tilde{G}_{Pi+1}^{(1,k)} \quad \Delta \tilde{G}_{Ii+1}^{(1,k)}) \quad (9)$$

$$[K_i^{(1,k)}] = \begin{bmatrix} [K_{PP}^{(1,k)}] & [K_{IP}^{(1,k)}] \\ [K_{IP}^{(1,k)}]^T & [K_{II}^{(1,k)}] \end{bmatrix} \quad (10)$$

Employing (7)-(10) we obtain the following relationships for the inner and perimeter nodes

$$\Delta \tilde{F}_{Pi+1}^{(1,k)} = [\kappa_{Pi}^{(1,k)}] \Delta \tilde{Y}_{Pi+1}^{(1,k)} + \Delta \tilde{f}_{Pi+1}^{(1,k)} \quad (11)$$

$$\Delta \tilde{Y}_{Ii+1}^{(1,k)} = -[\kappa_{PIi}^{(1,k)}] \Delta \tilde{Y}_{Pi+1}^{(1,k)} + \Delta \tilde{f}_{Ii+1}^{(1,k)} \quad (12)$$

where

$$[\kappa_{\text{PPI}}^{(1,k)}] = [K_{\text{PPI}}^{(1,k)}] - [K_{\text{PII}}^{(1,k)}]^T [K_{\text{III}}^{(1,k)}]^{-1} [K_{\text{PII}}^{(1,k)}] \quad (13)$$

$$\Delta_{\text{PII}}^{f(1,k)} = [K_{\text{PPI}}^{(1,k)}] [K_{\text{II}}^{(1,k)}] (\Delta_{\text{III}}^{F(1,k)} - \Delta_{\text{III}}^{G(1,k)}) + \Delta_{\text{PII}}^{G(1,k)} \quad (14)$$

$$[\kappa_{\text{PII}}^{(1,k)}] = [K_{\text{II}}^{(1,k)}]^{-1} [K_{\text{PII}}^{(1,k)}] \quad (15)$$

$$\Delta_{\text{III}}^{f(1,k)} = [K_{\text{III}}^{(1,k)}]^{-1} (\Delta_{\text{III}}^{F(1,k)} - \Delta_{\text{III}}^{G(1,k)}) \quad (16)$$

Assembling (11) yields the second-level substructural relationships, namely

$$\Delta_{\text{i}}^{F(2,k)} = [K_{\text{i}}^{(2,k)}] \Delta_{\text{i+1}}^{Y(2,k)} + \Delta_{\text{i+1}}^{G(2,k)} \quad (17)$$

$k = 1, 2, \dots$ Number of second level substructure

By partitioning (16) into inner and perimeter degrees of freedom we yield the third-order substructural relations after the appropriate assembly process. Continuing the partitioning and assembly process yields the various higher order substructural relations specifically

$$\Delta_{\text{i+1}}^{F(j,k)} = [K_{\text{i}}^{(j,k)}] \Delta_{\text{i+1}}^{Y(j,k)} + \Delta_{\text{i+1}}^{G(j,k)} \quad (18)$$

wherein the associated inner and perimeter partitions take the form

$$\Delta_{\text{PII}}^{f(j,k)} = [\kappa_{\text{PI}}^{(j,k)}] \Delta_{\text{PII}}^{Y(j,k)} + \Delta_{\text{PII}}^{f(j,k)} \quad (19)$$

$$\Delta_{\text{III}}^{Y(j,k)} = -[\kappa_{\text{PII}}^{(j,k)}] \Delta_{\text{PII}}^{Y(j,k)} + \Delta_{\text{III}}^{f(j,k)} \quad (20)$$

such that

$$[\kappa_{PI}^{(j,k)}] = [K_{PPI}^{(j,k)}] - [K_{PII}^{(j,k)}]^T [K_{III}^{(j,k)}]^{-1} [K_{PII}^{(j,k)}] \quad (21)$$

$$\Delta f_{PI+1}^{(j,k)} = [K_{PII}^{(j,k)}] [K_{III}^{(j,k)}] (\Delta F_{II+1}^{(j,k)} - \Delta G_{II+1}^{(j,k)}) + \Delta G_{PI+1}^{(j,k)} \quad (22)$$

$$[\kappa_{PII}^{(j,k)}] = [K_{III}^{(j,k)}]^{-1} [K_{PII}^{(j,k)}] \quad (23)$$

$$\Delta f_{II+1}^{(j,k)} = [K_{III}^{(j,k)}]^{-1} \Delta F_{II+1}^{(j,k)} \quad (24)$$

Based on (11)-(24), we see that the overall nonlinear hierarchical substructuring requires a forward calculation phase as well as a backward stage. The forward phase involves the use of (11), (13), (14), (19), (21) and (22). In contrast the backward phase, which involves the definition of inner nodes, incorporates the use of (12), (15), (16), (20), (23) and (24). In terms of the forward iterative algorithms, the overall required machine architecture takes the form defined in Figure 6. Note the common data buses linking successive substructural levels need only provide access to perimeter data. In this way, significantly less data need to be accessed by the global-level DBM. This applies throughout the forward phase of the iteration process. Overall the steps handled by each of the succeeding levels involve assembly, inner/perimeter partitioning, and setting up effective stiffnesses for the forward and backward phases. In terms of (21)-(24), the stiffnesses associated with the perimeter and inner nodes involve an inverse of the inner partition of the k^{th} substructural stiffness. All such manipulations must be performed by processors dedicated to each of the k individual substructures associated with the various hierarchical levels.

Once the forward loop of calculations is complete, the perimeter data must be back tracked to the inner nodes of each of the various substructures at the different substructural levels. The overall flow of control/calculation is depicted in Figure 7. As can be seen, the perimeter data are used to determine the inner nodal incremental excursions. This is achieved through the use of the family of expressions defined by (20). Once the back substitutions to the succeeding levels up to and including the first are completed, the standard norm type convergence checks must be implemented to ascertain the quality of convergence. Contingent on the convergence check, the iteration process can be cycled through the forward and backward phases of the substructural hierarchy.

6. Discussion

To illustrate the hierarchical substructural scheme, consider the three-level simulation defined in Figure 8. The number of nodes and substructure associated with the example are given in Figure 9. Based on the number of inner and perimeter variables depicted, the expressions defining the number of respective nodes are given by:

i) Level 1

$$\text{Perimeter Nodes} = 2(\ell_1 + \ell_2 + \ell_3) \quad (25)$$

$$\text{Inner Nodes} = (\ell_1 - 2)(\ell_2 - 2) \quad (26)$$

ii) Level 2

$$\text{Perimeter Nodes} = 2(\ell_1 n_1 + \ell_2 n_2 - n_1 - n_2) \quad (27)$$

$$\text{Inner Nodes} = n_1 n_2 (\ell_1 + \ell_2) - n_1 (\ell_1 + 2) - n_2 (\ell_2 + 2) - 3n_1 n_2 + 1 \quad (28)$$

iii) Level 3

$$\text{Perimeter Nodes} = 2m_1 n_1 (\ell_1 - 1) + 2m_2 n_2 (\ell_2 - 1) \quad (29)$$

$$\begin{aligned} \text{Inner Nodes} = & \ell_1 m_1 n_1 (m_2 - 1) + \ell_2 n_2 m_2 (m_1 - 1) - m_1 m_2 (n_1 + n_2) \\ & + m_1 n_1 + m_2 n_2 - m_1 m_2 + 1 \end{aligned} \quad (30)$$

Employing (25-30) we see that the storage effectiveness of each of the various levels is expressed by the relations

$$\xi^{(1)} = \frac{\text{Perimeter}}{\text{Perimeter} + \text{Inner}} \quad (31)$$

where k denotes the level number. In the context of (31), it follows that

$$\xi^{(1)} = \frac{2(\ell_1 + \ell_2 - 2)}{\ell_1 \ell_2} \quad (32)$$

$$\xi^{(2)} = \frac{2(\ell_1 n_1 + \ell_2 n_2 - n_1 - n_2)}{n_1 \ell_1 (n_2 + 1) + \ell_2 n_2 (n_1 + 1) - 3n_1 n_2 - 4(n_1 + n_2) + 1} \quad (33)$$

$$\xi^{(3)} = \frac{2m_1 n_1 (\ell_1 - 1) + 2m_2 n_2 (\ell_2 - 1)}{\ell_1 m_1 n_1 (m_2 + 1) + \ell_2 m_2 n_2 (m_1 + 1) - m_1 m_2 (n_1 + n_2 + 1) - m_1 n_1 - m_2 n_2 + 1} \quad (34)$$

Consider the case wherein

$$\ell_1 = 100, \ell_2 = 50$$

$$n_1 = 5, n_2 = 4$$

$$m_1 = 3, m_2 = 4$$

In terms of the foregoing, Table 2 gives the total number of

- degrees of freedom
- processors required at each level
- perimeter/inner nodes

as well as the storage effectiveness of each of the substructural levels. Noting that a straight solution of the given problem would require a 1.2×10^6 order stiffness matrix, it follows from Table 2 that very significant storage savings as well as speed enhancements can be achieved.

In the context of the foregoing development, it follows that hierarchical substructural parallelism has decided advantages over vector-type processors, namely:

- i) Global common core is reduced in size
- ii) Substructures are handled in smaller local cores which could employ vector processors and which are controlled by local DBM
- iii) Data transfer between succeeding levels of substructural hierarchy are reduced thereby reducing load on DBM
- iv) Various substructures are updated, inverted, and assembled simultaneously hence enhancing the overall speed
- v) The overall addressing requirements are reduced since the size of individual substructural zones is much smaller
- iv) Extensive use of cash memory (Ram Disk) can be made at the local level thereby reducing disk I/O
- vii) Backward and forward steps follow natural formulational lines
- viii) Element-to-element or hierarchical least-squares algorithms can be employed at the local substructural level
- ix) Linear/nonlinear problem partitioning can be more logically handled
- x) Overall control of the machine is more logical and less difficult since local processors are essentially autonomous within updating and inverting phases of the operation

- xi) The MCL can be patterned about well-defined substructuring methodology; the transfer of control from level to level is contingent on the monitoring status of stiffness/inversion calculations
- xii) The data base manager needs only to deal with data residing on perimeters of the substructure; as noted earlier, this significantly reduces the amount of data transferred between levels.

As discussed earlier, the modeling of tires in their use environment represents perhaps one of the most comprehensive single component nonlinear structural response problems currently available. This follows from the fact that geometric-, material-, and boundary-induced nonlinearity all simultaneously act to define the global response behavior. Due to their regionalized/ substructural form of construction, tires represent a good modeling problem to help define the architecture of high-level multiprocessor machines. In this context, a hierarchical form of substructural parallelism has decided advantages over other forms of multiprocessors. As has been seen such a procedure has several theoretical advantages for nonlinear problems. These evolve about the simplified DBM structure, reduced data flow, smaller global core, and reduced addressing requirements.

Overall future work in this area should

- Place greater emphasis on algorithmic architecture and its possible effects on machine structure
- Establish proper control configuration for hierarchical DBM
- Extend scheme to constrained incremental Newton/Raphson (INR) least-squares algorithms as well as transient schemes
- Apply concept to available parallel processors
- Structure procedure so as to enable either direct or iterative solutions at substructural level
- Establish criteria to enable determination of quality of convergence at local substructural level

6. References

1. Zienkiewicz, O.C., The Finite Element Method, McGraw Hill Company (1983).
2. Bathe, K.J., Finite Element Procedures in Engineering Analysis, Prentice Hall (1983).
3. Oden, J.T., Finite Elements of Nonlinear Continuum, McGraw Hill Co. (1972).
4. Cook, R.D., Concepts and Applications of Finite Element Analysis, Wiley (1983).
5. Hughes, T.J.R., Winget, J. and Levit, I., Comp. Meth. Appl. Mech. Engineering, 36, 241 (1983).
6. Padovan, J. and Arechaga, T., Int. Jr. Engineering Science, 20, 1077 (1982).
7. Padovan, J. and Lackney, J., Computers and Structures, 19, 535 (1984).
8. Noor, A.K., Storaasli, O.O. and Fulton, R.E., ASME Special Publ., H00275, 1 (1983).
9. Bostic, S.W. and Fulton, R.D., AIAA Paper No. 85-0783-CP (1985).
10. Padovan, J. and Tovichakchaikul, S., Computers and Structures, 18, 191 (1984).
11. Padovan, J., Global-Local Finite Element Modeling, 126th Meeting of the ACS Rubber Division (1984).
12. Clark, S., Mechanics of Pneumatic Tires, U.S. Department of Transportation (1980).
13. Padovan, J. and Nakajima, Y., Numerical Simulation of Tire Skidding Events Involving Impacts with Holes and Bumps, 4th Annual Tire Society Meeting (1985).
14. Padovan, J. and Nakajima, Y., Numerical Simulation of Tire Transiently Sliding/Rolling Over Holes and Bumps, ACS Meeting (April 1985).
15. Padovan, J. and Paramadilok, O., Transient and Steady State Viscoelastic Rolling Contact, Computers and Structures, 20, 545-553 (1985).
16. Trinko, J.J., Ply and Rubber Stresses and Contact Forces for a Loaded Radial Tire, First Annual Meeting of Tire Society, Akron (1982).
17. Tabaddor, F. and Stafford, J., Computers and Structures, 13, 737 (1981).

TABLE 1 COMPARISON OF LINEAR AND NONLINEAR FE SIMULATION OF PRESSURIZED TIRE

PRESSURE (PSI)	LINEAR (IN) MAXIMUM DEFLECTION		NONLINEAR (IN) MAXIMUM DEFLECTION	
	CROWN	SIDEWALL	CROWN	SIDEWALL
5.	.003	.036	.003	.027
10.	.006	.073	.006	.047
15.	.009	.109	.009	.064
30.	.018	.218	.018	.107

TABLE 2 COMPARISON OF HIERARCHICAL SUBSTRUCTURAL PARALLEL AND SINGLE PROCESSOR SYSTEMS

LEVEL	HIERARCHICAL SUBSTRUCTURAL PARALLEL SYSTEM					SINGLE PROCESSOR D of F
	NUMBER OF PROCESSORS	PERIMETER D of F/P*	INNER D of F/P	TOTAL D of F/P**	STORAGE EFF.	
1	240	296	4704	5000	.0592	---
2	20	1382	2223	4605	.3	---
3	12	4538	6012	10550	.3	---
GLOBAL	---	---	---	---	---	1200000

*D of F/P - degrees of freedom per processor

**Total number of perimeter and inner D of F

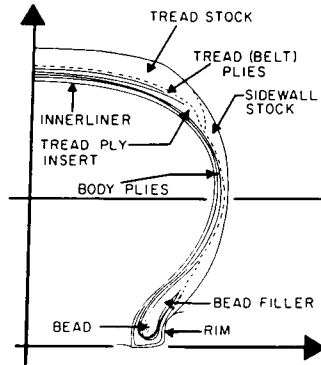
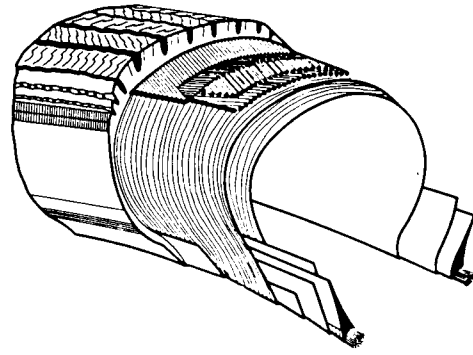


Figure 1 Tire geometry and cross section.

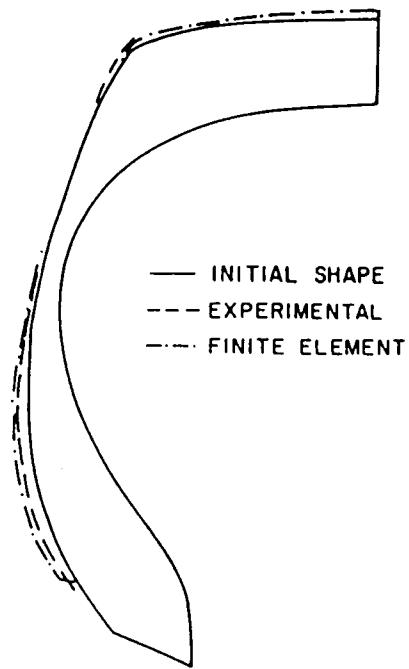


Figure 2 Pressure tire profile.

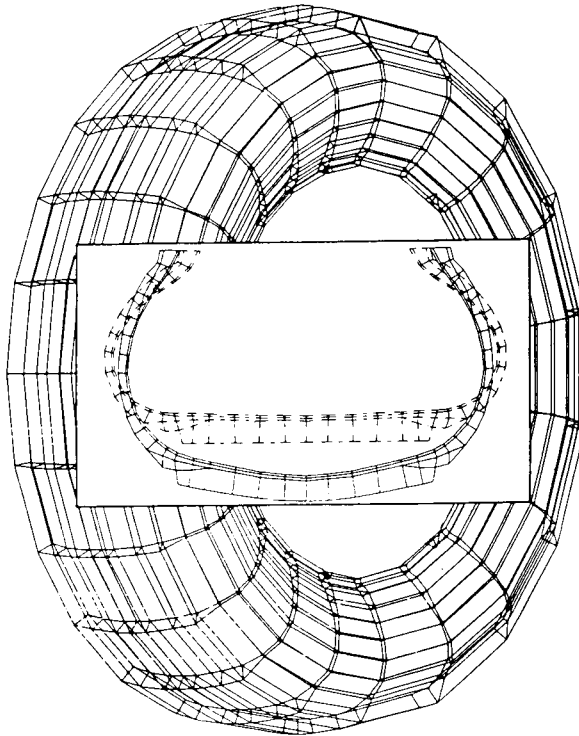


Figure 3 Loading into standing contact.

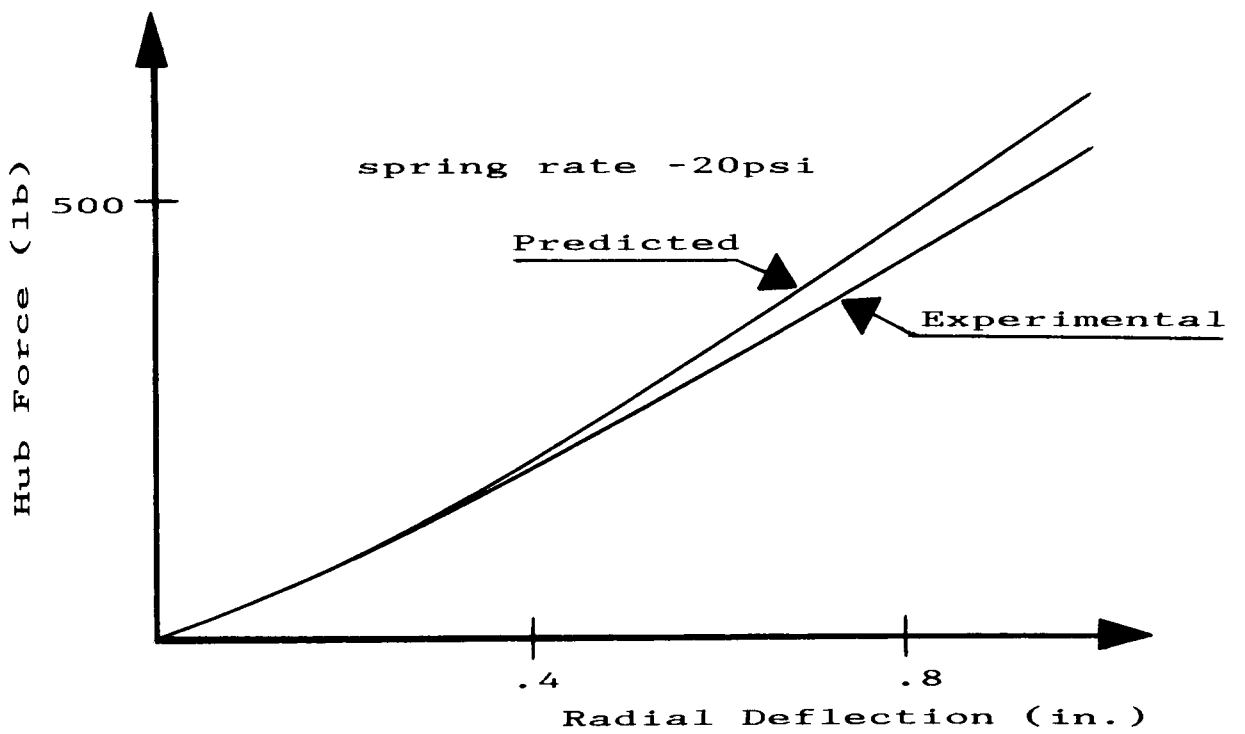


Figure 4 Force deflection characteristics into standing contact.

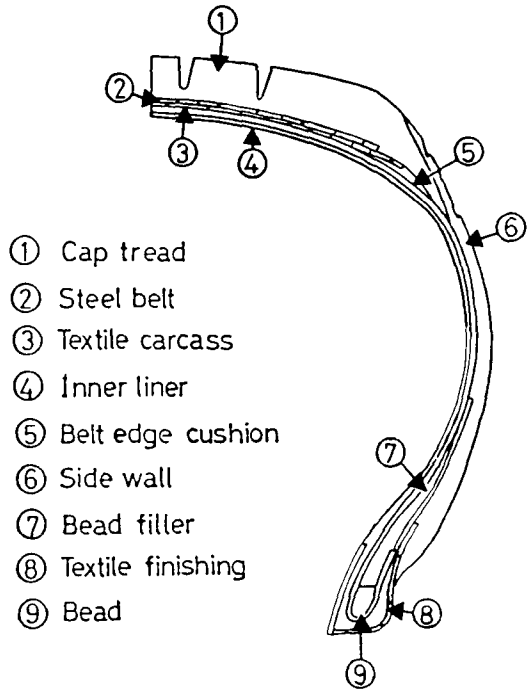


Figure 5 Substructural zones of tire.

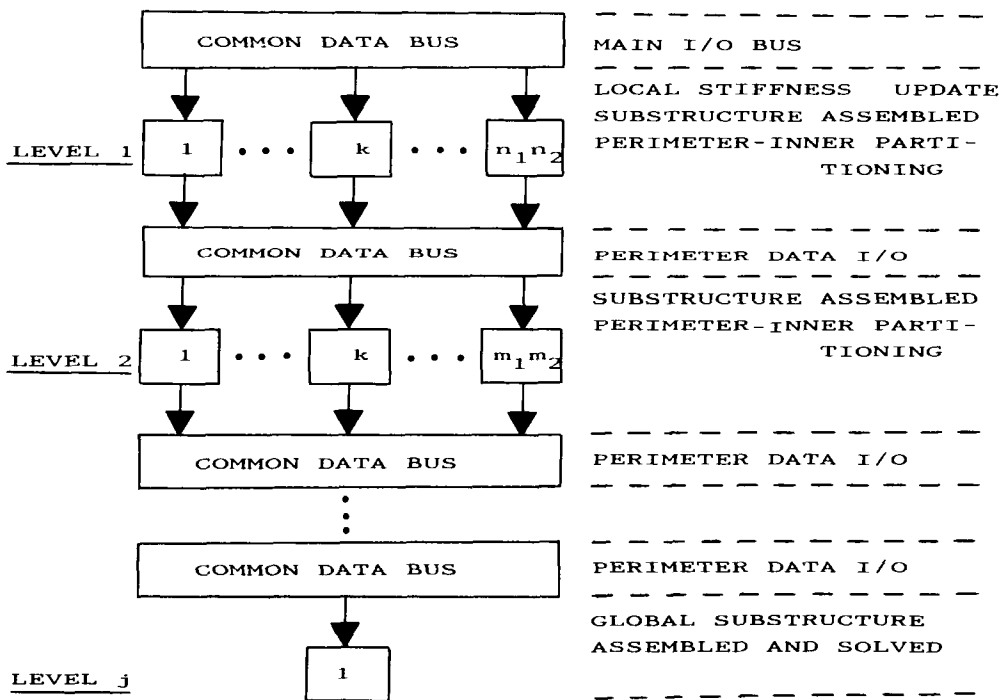


Figure 6 Flow of control: forward loop.

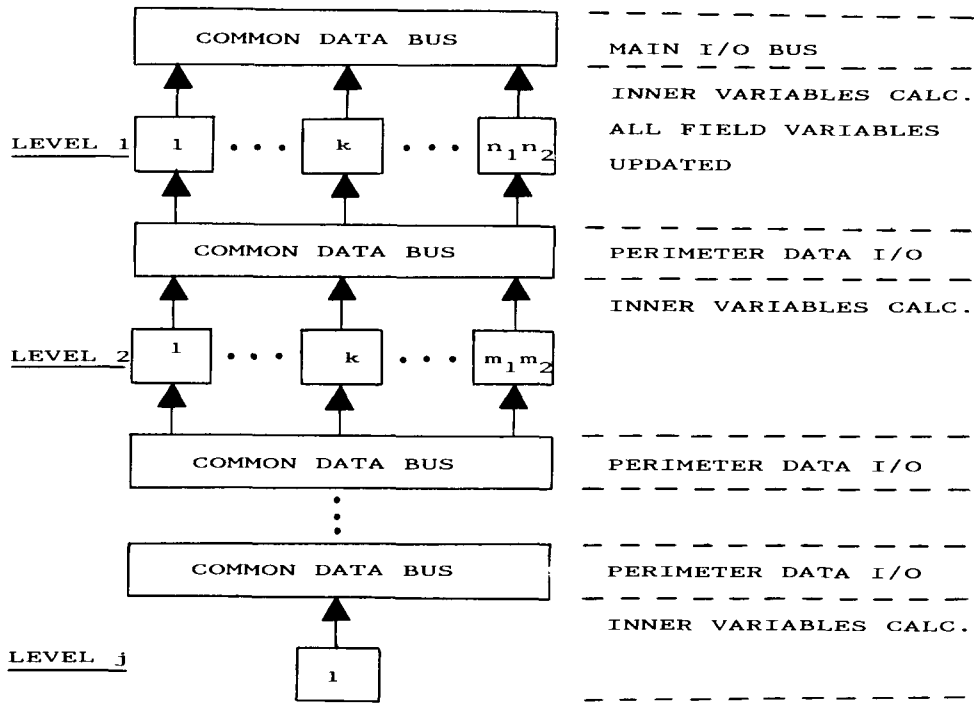


Figure 7 Flow of control: backward loop.

- LEVEL 1 (LOCAL)
- LEVEL 2 (INTERMEDIATE)
- - - LEVEL 3 (GLOBAL)

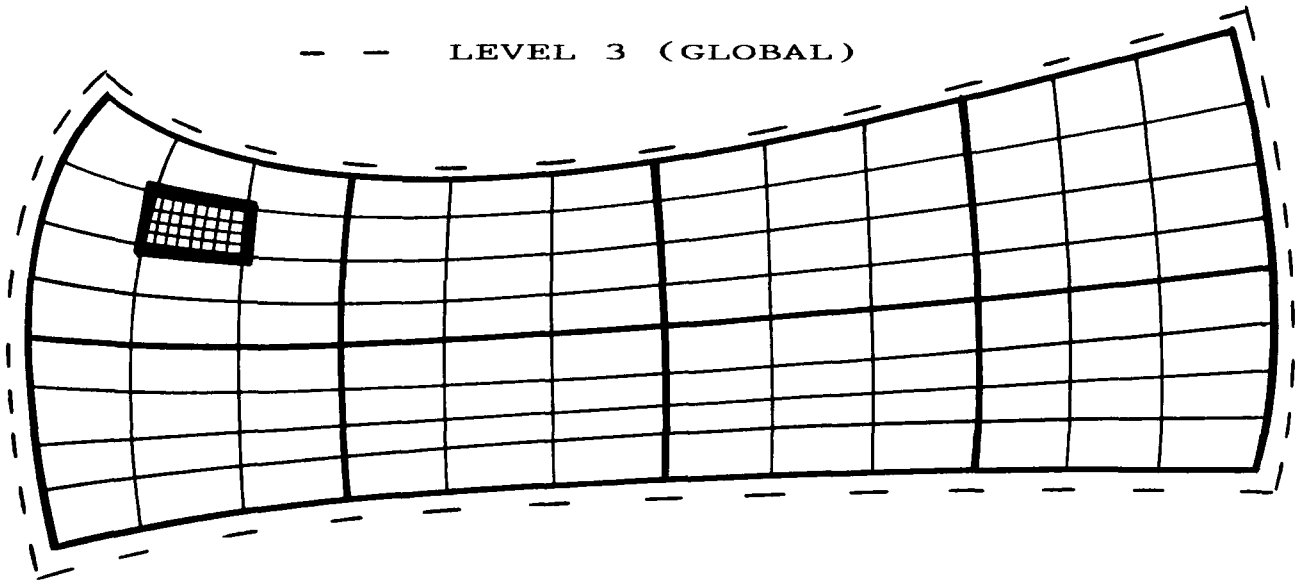


Figure 8 Example of three-level hierarchical substructural system.

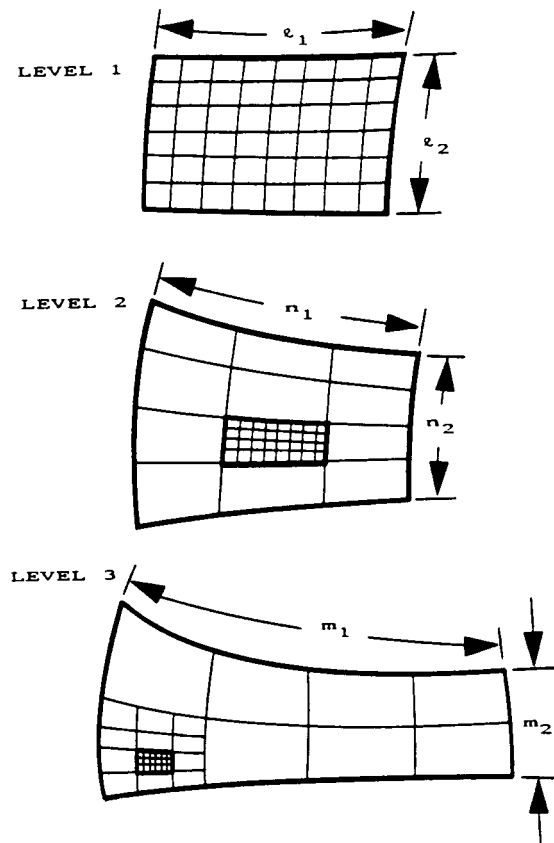


Figure 9 Number of perimeter substructures at each level.

**ADAPTIVE METHODS, ROLLING CONTACT,
AND NONCLASSICAL FRICTION LAWS**

J. T. Oden
The University of Texas at Austin
Austin, Texas

ABSTRACT

Results and methods on three different areas of contemporary research are outlined. These include adaptive methods, the rolling contact problem for finite deformation of a hyperelastic or viscoelastic cylinder, and non-classical friction laws for modeling dynamic friction phenomena.

1. INTRODUCTION

This paper addresses three subjects that impact on the computer simulation of nonlinear tire behavior: adaptive methods, which represent schemes for assessing numerical error and automatically adapting the mesh so as to improve accuracy; the rolling contact problem, which is at the heart of tire analysis; and new friction laws, which are essential in developing realistic models of frictional contact. Space limitations preclude a detailed discussion of these issues; but further details can be found in recent papers and reports by the author and his colleagues [1-17].

2. ROLLING CONTACT

The general rolling contact problem as a basis for nonlinear tire analysis involves some of the most challenging and difficult problems in structural mechanics. Among the complicating features are the presence of unilateral contact, friction, inertia effects, multi-parameter bifurcations, the emergence of standing waves, viscoelastic and thermal effects, large deformations, the necessity of modeling of complex materials such as fiber-reinforced rubbers, and the presence of non-conservative pressurization loadings. A first step toward resolving such issues is the formulation of correct kinematics and variational principles for a special case: the steady-state rolling of a hyperelastic or viscoelastic cylinder in contact with a rigid foundation and in a state of plane strain.

The kinematical situation is shown in Fig. 1 where the geometry of the reference configuration (a rigid spinning cylinder with no contact) is compared to the geometry of a deformed cylinder in steady-state rolling contact with a rough (frictional) roadbed (foundation).

Key features of the kinematic description are listed as follows:

1) Time appears only implicitly in the formulation; if (R, Θ, Z) are material coordinates, the referential coordinates are

$$r = R, \quad \theta = \Theta + \omega t, \quad z = Z$$

where ω is the angular velocity of the rigid, reference cylinder.

2) If the motion is defined by the map

$$x_i = \chi_i(r, \theta, z) \quad i = 1, 2, 3$$

where x_i = spatial Cartesian coordinates of particles in the current configuration, then velocity and acceleration components are

$$v_i = \omega \frac{\partial \chi_i}{\partial \theta} \quad a_i = \omega^2 \frac{\partial^2 \chi_i}{\partial \theta^2}$$

3) The unilateral contact constraint can be expressed in the form

$$\chi_2 \leq H \text{ on } \Gamma_C$$

where Γ_C is the exterior contact surface and H is the distance from the hub center to the foundation. This condition can also be written

$$(\chi_2 - H)_+ = 0$$

where $(\cdot)_+$ denotes the positive part of (\cdot) .

4) The time history of deformation can be expressed in terms of strains of particles located on the same circular arc in the reference configuration. For example, if \underline{E} is the Green-Saint Venant strain tensor, its time history over an interval $0 \leq \tau \leq t$ satisfies:

$$\{\underline{E}(r, \theta, z, \tau); 0 \leq \tau \leq t\} = \{\underline{E}(r, \chi, z, t), 0 \leq \chi \leq \omega t\}$$

This property makes it possible to incorporate viscoelastic effects into the rolling contact problem in a straightforward way.

For illustration purposes, we consider a class of rolling contact problems in which the following constitutive properties hold:

a) The material is either an isotropic hyperelastic material characterized by a strain energy function

$$W = W(I_1, I_2, I_3)$$

(or $W = W(I_1, I_2)$ if $I_3 = 1$ - an incompressible material) in which I_1, I_2, I_3 are the principal invariants of the deformation tensor $\underline{C} = \underline{F}^T \underline{F}$, or the material is a viscoelastic material characterized by a linear viscoelastic perturbation of a hyperelastic material; e.g.,

$$\underline{S} = \frac{\partial W}{\partial \underline{E}} + \gamma \int_{-\infty}^t k(\tau, t) \dot{\underline{E}}(\tau) d\tau$$

with γ a viscosity parameter, k a material kernel, and \underline{S} the second Piola-Kirchhoff stress tensor.

b) The normal stiffness of the contact interface obeys a constitutive law of the type

$$t_n = c_n (\chi_2 - H)^{m_n} \text{ on } \Gamma_C$$

where t_n is the normal stress and c_n and m_n are material constants. If $m_n = 1$ and $c_n = 1/\epsilon$, where ϵ is a positive constant, this relation coincides with the normal contact stress associated with an exterior penalty approximation of the unilateral constraint condition.

c) If the cylinder is given a prescribed velocity v_0 relative to the roadway, the slip velocity on the contact surface is

$$w_T = v_0 - \dot{\chi}_1 = v_0 - \omega \partial_\theta \chi$$

5) The friction law is (with $t_{\sim T}$ the frictional stress)

$$|t_{\sim T}| < \mu |t_n| \Rightarrow w_T = 0$$

$$|t_{\sim T}| = \mu |t_n| \Rightarrow w_T = -\lambda |t_{\sim T}| \text{ for some } \lambda \geq 0$$

Variational principles for various rolling contact problems are summarized in Figs. 2-8, beginning with the pure spinning of a cylinder without contact and progressing to the general variational inequality for finite deformation rolling contact with friction. Various spaces of admissible functions are defined in these figures as well as several nonlinear forms. In particular,

- $A(\underset{\sim}{\chi}, \underset{\sim}{\eta})$ = the internal virtual work produced by the Piola-Kirchhoff stresses T_0
- $B(\underset{\sim}{\chi}, \underset{\sim}{\eta})$ = the virtual work produced by inertia (radial acceleration) effects per unit of angular velocity ω
- $C(\underset{\sim}{\chi}, \underset{\sim}{\eta})$ = work term due to normal compliance of the interface
- $I(\underset{\sim}{\chi}, \underset{\sim}{\xi}, \underset{\sim}{\eta})$ = a virtual work term representing the work done by the hydrostatic pressure p (present when the material is incompressible)
- $j(\underset{\sim}{\chi}, \underset{\sim}{\eta})$ = the virtual work term due to frictional forces
- $f(\underset{\sim}{\eta})$ = the virtual work due to external forces

A finite-element code has been developed based on this general variational principle which has the following features:

1. Biquadratic (Q_2) elements are used to approximate the displacement field and, for incompressible materials, P_1 , discontinuous linear elements are used to approximate hydrostatic pressures
2. The frictional functions are regularized in a standard way
3. A Riks-Crisfield method with Newton-Raphson corrections is used to solve the nonlinear systems of equations characterizing the discrete problem

To date, an extensive set of numerical solutions has been obtained using these concepts and methods. Here only one example is cited, which is interesting because of the slow emergence of standing waves as the angular velocity is increased for a fixed penetration H of a hollow rubber cylinder into a rough roadway with coefficient of friction $\mu = 0.03$. Computed deformed shapes and stress contours are shown in Figs. 9-13 for various values of ω . We notice the steady development of more-or-less periodic wavelets on the exterior surface which meet at points at which singularities appear to be forming. The presence of friction on the contact surface destroys the symmetry of this wavelet pattern. Mild viscoelastic effects, such as those in rubbers at moderate temperatures, do not appreciably alter the structure of these deformed geometries.

The generality of the formulation and of the methods employed here makes it possible to study numerically a wide range of rolling contact problems. Further work shall involve generalizations of these results to three-dimensional rolling contact problems which include the effects of turning, steering forces, and tilting relative to the roadway plane.

3. ADAPTIVE METHODS

We shall now turn to the important subject of adaptive finite-element methods. Adaptive methods should have a significant impact on not only tire analysis but also on general computational structural mechanics in the relatively near future.

In general, adaptive methods seek to change the structure of an approximate method to improve the quality of the solution. By structure, we mean the mesh density, locations of nodes, and order of the local polynomials. By quality of an approximation, we generally mean the error in approximation in some appropriate norm. There are, thus, two primary aspects of any adaptive finite-element method:

- 1) The estimation of the error
- 2) The adaptive strategy

In the first of these, it is assumed that an initial approximation of the solution is known, perhaps from a computation on a coarse mesh, and that this rough solution can be used to obtain an a posteriori estimate of the local

error over each finite element. Once an estimate of the local error is known, one must call upon some technique to reduce the local error and thereby improve the quality of the solution.

There are two general types of methods we have studied for a posteriori error estimation of the local error over each finite element. Once an estimate of the local error is known, one must call upon some technique to reduce the local error and thereby improve the quality of the solution.

There are two general types of methods we have studied for a posteriori error estimation:

1. Residual methods
2. Interpolation (or a priori) methods

As the name implies, residual methods make use of element residuals - the residual or unbalance left over when the approximate solution is substituted into the governing equations and edge conditions on each element.

The residual itself (e.g., the equilibrium unbalance in element forces) is not necessarily a good indication of local error. Indeed, the local residual can be nearly zero while the error can be quite large. For this reason, it is generally necessary to calculate certain local error indicators ϕ_K which bound the error above and below in appropriate norms. The calculation of error indicators generally requires the solution of special local problems over each element in which the element residuals enter as data. For example, in the model elliptic problem,

$$-\Delta u = f \text{ in } \Omega \subset \mathbb{R}^2$$

$$u = 0 \text{ on } \partial\Omega$$

(with Δ the Laplacian and f given), the finite-element solution u_h satisfies

$$\int_{\Omega} \nabla u_h \cdot \nabla v_h \, dx dy = \int_{\Omega} f v_h \, dx dy$$

for arbitrary test function v_h , and over each element K of a mesh, the residual is

$$r_h = -\Delta u_h - f$$

Over element K , an error indicator ϕ_K is computed which satisfies

$$\int_K \nabla \phi_K \cdot \nabla v \, dx dy = \int_K r_h v \, dx dy + \phi_K \frac{\partial u_h}{\partial n} v ds$$

for v in $H^1(K)$. One can show that the error $e_h = u - u_h$ in the energy norm ($\|e_h\|_{1,\Omega} = \left\{ \int_{\Omega} |\nabla e_h|^2 \, dx dy \right\}^{1/2}$) satisfies the bound

$$\|e_h\|_{1,\Omega}^2 \leq \sum_K \|\phi_K\|_{1,K}^2$$

Various residual methods differ in the way these error indicators are defined and calculated. There are some residual techniques which can produce sharp local error estimates in virtually any norm for certain classes of problems. (See Demkowicz and Oden [4, 5]). These methods are not restricted to linear problems and have been used to produce error estimates in highly non-linear problems (see [7, 16]).

The interpolation methods make use of the fact that the interpolation error over an element K of diameter h_K over an element on which polynomials of degree p are used is

$$|u - \Pi_h u|_{1,K}^2 \leq C h_K^{2p} |u|_{2,K}^2$$

where u is a given smooth function, $\Pi_h u$ is its interpolant,

$$|u|_{1,K}^2 = \int_K |\nabla u|^2 \, dx dy$$

$$|u|_{2,K}^2 = \int_K (u_{xx}^2 + u_{xy}^2 + u_{yy}^2) \, dx dy$$

and C is a constant independent of h_K and u . The idea behind interpolation methods is to estimate $|u|_{2,K}^2$ using results of a coarse-mesh approximation (e.g., using finite-difference methods or extraction methods [6]). The constant C cannot, in general, be determined, so such interpolation methods can only be used to assess relative error in various finite elements.

Once an estimate of the error is obtained, the local error is reduced adaptively using one of the following techniques:

1. h-methods: the mesh size h is reduced and the number of elements is increased in regions of high error.

2. p-methods: the mesh is fixed, but the local order p of the polynomial shape functions is increased.

3. Moving mesh methods: the nodes in a finite-element mesh are moved and concentrated in areas of high error.

4. Combined methods: these involve combinations of the above three techniques.

We have developed test codes which employ all of these methods. The results of some tests are given in Figs. 14-20, and specific comments follow.

1. In Fig. 14 we see a distorted mesh obtained using a moving mesh strategy on the driven cavity problem for an incompressible viscous fluid (see [7]).

2. Figures 15, 16, and 17 contain computed results in adaptive schemes based on residual methods devised by Demkowicz and Oden [4, 5]. The results shown here are for transient heat conduction problems with dominate convection effects and for nonlinear Burgers' equation vector-valued solutions which simulate the Navier-Stokes equations. A special h-method is used here which employs a fine grid and a coarse-grid approximation to estimate error.

3. Results of a new p-method for Navier-Stokes equations in two-dimensions are illustrated in Figs. 18, 19 (see [1, 16]). Here a rather coarse mesh is used and errors are reduced by increasing local polynomial degrees from 1 to 2 to 3. Different shading in these figures indicates different levels of local L^2 error, with black cells indicating an error of less than 5 percent, grey an error of less than 10 percent, and white an error of over 20 percent. Such large local errors are reduced before the solution is allowed to advance in time. Remarkably, the so-called effectivity index θ for this problem (which represents the ratio of the estimated local error to the actual local error) for an L^2 - norm varied from around 1.001 to 0.860 for the time ranges considered in a test example. This suggests that residual-type error estimates based on p-type strategies can be very accurate, even for transient nonlinear problems on coarse meshes.

Figure 20 shows refined mesh patterns for a class of linear elliptic problems in which a very fast vectorizable h-method is used in conjunction with an interpolation-type error estimator (see [6]). One interesting aspect of this work, indicated by different shadings of elements in the figure, is that the distinction between "optimal" meshes determined using a very sophisticated error estimator (see [17]) and very crude estimates ([6]) is negligible whenever strong singularities are present.

4. NON-CLASSICAL FRICTION LAWS

In our most recent calculations of rolling contact, we have employed special interface constitutive equations for the normal compliance of the

interface and the tangential frictional forces. Some of these laws are mentioned in Section 2 of this paper (see also Fig. 6). The physical interpretation and the motivation of such models are discussed in references [14, 15, and 18].

ACKNOWLEDGMENT

The work reported here was supported by several agencies: the work on adaptive methods by The Office of Naval Research under Contract N00014-84-K-0409; the work on rolling contact by Contract NAS1-17359 from the NASA Langley Research Center, and work on non-classical friction by the Air Force Office of Scientific Research under Contract F49620-84-0024.

REFERENCES

1. DEMKOWICZ, L., ODEN, J.T., STROUBOULIS, T. and CAMPOS L., Variational Principles and Adaptive Schemes for a Class of Flow Problems with Moving Boundaries, Finite Elements in Flow Problems, Vol. VI, John Wiley and Sons, Ltd., London, 1985.
2. ODEN, J.T., and DEMKOWICZ, L., A Posteriori Error Estimates and Adaptivity in Finite Element Methods, Integral Methods in Science and Engineering, Hemisphere Publishers, Washington, 1986.
3. ODEN, J.T., and DEMKOWICZ, L., Adaptive Finite Element Methods for Complex Problems in Solid and Fluid Mechanics, International Conference on Finite Elements in Computational Mechanics, 2-6 December, 1985, Bombay India.
4. DEMKOWICZ, J., and ODEN, J.T., An Adaptive Characteristic Petrov-Galerkin Finite Element Method for Convection-Dominated Linear and Nonlinear Parabolic Problems in One Space Variables, Texas Institute for Computational Mechanics, TICOM Report 85-3, Austin, TX, June, 1985.
5. DEMKOWICZ, L., and ODEN, J.T., An Adaptive Characteristic Petrov-Galerkin Finite Element Method for Convection-Dominated Linear and Nonlinear Parabolic Problems in Two Space Variables, TICOM Report 85-7, 1985.
6. DEMKOWICZ, L., DEVLOO, P.H., and ODEN, J.T., On an H-Type Mesh Refinement Strategy Based on Minimization of Interpolation Errors, Computer Methods in Applied Mechanics and Engineering. Vol. 53, pp. 67-89, 1985.
7. DEMKOWICZ, L., and ODEN J.T., On a Mesh Optimization Method Based on Minimization of Interpolation Error, Int. J. Eng. Sci., Vol. 24, No. 1, pp. 55-68, 1986s.
8. ODEN J.T., DEMKOWICZ, L., and STROUBOULIS, T., Adaptive Finite Element For Flow Problems With Moving Boundaries. I. Variational Principles and A Posteriori Estimates, Computer Methods in Applied Mechanics and Engineering, Vol. 46, pp. 217-251, 1984.
9. DEMKOWICZ, L., and ODEN J.T., An Adaptive Characteristic Petrov-Galerkin Finite Element Method For Convection-Dominated Linear and Nonlinear Parabolic Problems in Two Space Variable, Computational Methods in Applied Mechanics, (to appear).
10. ODEN J.T., DEMKOWICZ, L., and STROUBOULIS, T., Adaptive Methods for Incompressible Viscous Flow Problems with Moving Boundaries, Accuracy Estimates and Adaptive Refinements in Finite Element Computations, John Wiley and Sons Ltd., London, (to appear), (Proceedings, Lisbon Portugal, June 19-22, 1984).
11. STROUBOULIS, T., ODEN J.T., and DEMKOWICZ, L., A Posteriori Error Estimates for Some Galerkin Space-Time Approximations of the Unsteady Navier-Stokes Equations, TICOM REPORT 85-5, JUNE 1985.

12. ODEN J.T., BECKER, E.B., LIN, T.L., and DEMKOWICZ, L., Formulation and Finite Element Analysis of a General Class of Rolling Contact Problems with Finite Elastic Deformation, Mathematics of Finite Elements with Applications, Academic Press Ltd., London, (to appear).
13. ODEN J.T., BECKER, E.B., LIN, T.L., and HSIEH, K.T., Numerical Analysis of Some Problems Related to the Mechanics of Pneumatic Tires: Finite Deformation of Cord-Reinforced Rubber Composites, Symposium on Advances and Trends in Structures and Dynamics, NASA CN, NASA Langley Research Center, (to appear). NASA CP 2335, Oct. 1984, pp 297-307.
14. ODEN J.T., and LIN, T.L., On the General Rolling Contact Problem for Finite Deformations of a Viscoelastic Cylinder, TICOM REPORT 85-, Austin, TX, 1985.
15. ODEN J.T., and MARTINS, J.A.C., Models and Computational Methods for Dynamic Friction Phenomena, Computer Methods in Applied Mechanics and Engineering, Vol. 52, pp. 527-634, 1985.
16. ODEN J.T., and DEMKOWICZ, L., Extraction Methods for Second-Derivatives in Finite Element Approximations of Linear Elasticity Problems, Communications in Applied Numerical Methods, Vol. 1, pp. 137-139, 1985.
17. MARTINS, J.A.C., and ODEN J.T., Interface Models, Variational Principles and Numerical Solutions for Dynamic Friction Problems, Mechanics of Material Interfaces, (Proceedings Session on Mechanics of Material Interfaces and Mechanics of Contact Problems, at the ASCE/ASME Mechanics Conference, Albuquerque, NM, June 24-26, 1985)

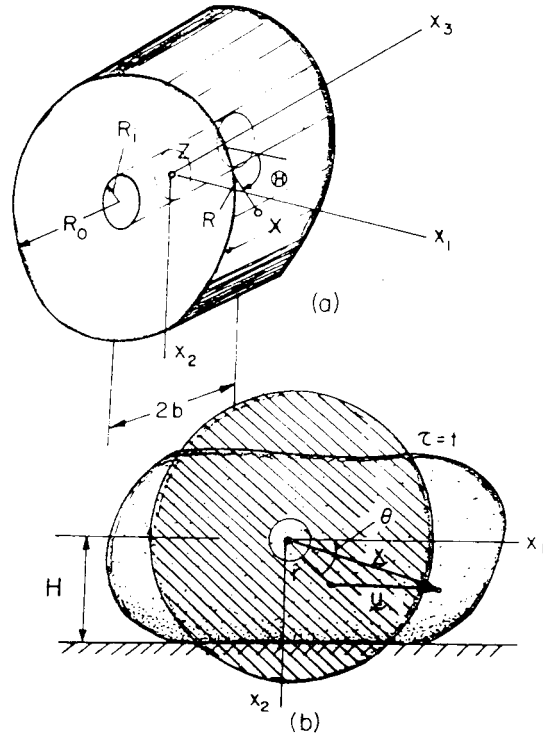


Figure 1

**STEADY SPINNING OF A
DEFORMABLE CYLINDER**

\mathcal{V} = SPACE OF ADMISSIBLE MOTIONS

$$\equiv \{ \eta \in V \mid \eta|_{(r,\theta)} = R_i \text{ a.e. on } \Gamma_D \}$$

$$V \equiv \{ \eta = (\eta_1, \eta_2) \mid \left| \int_{\Omega_0} W(\nabla \eta) \, dv_0 \right| < \infty \}$$

CASE: W = POLYNOMIAL OF I_C, II_C, III_C

V = SOBOLEV SPACE OF ORDER $(1, p)$

$$= (W^{1,p}(\Omega_0))^2 \quad \text{for some } p, \quad 2 \leq p < \infty$$

CASE: FOR VISCOELASTIC MATERIAL

$$U \equiv V \cup \{ \eta \in V \mid$$

$$\left| \int_{\Omega_0} F_{\tau=-\infty}^t(\tau, E(\nabla \eta)) : \nabla \eta \, dv_0 \right| < \infty \}$$

Figure 2

VARIATIONAL PROBLEM ≡ B.V.P

VARIATIONAL PROBLEM

FIND A MOTION $\chi \in \psi$ SUCH THAT

$$A(\chi, \eta) = \omega^2 B(\chi, \eta) + f(\eta) \quad \text{for all } \eta \in \psi$$

BOUNDARY VALUE PROBLEM

$$\text{Div} T_0(\chi) + \rho_0 b_0 = \rho_0 \partial_\theta^2 \chi \quad \text{in } \Omega_0$$

$$\chi = R_i \quad \text{on } \Gamma_D$$

$$T_0(\chi) n_0 = 0 \quad \text{on } \Gamma/\Gamma_D = \Gamma_C$$

=====

$$A(\chi, \eta) = \int_{\Omega_0} (\partial W(\nabla \chi) / \partial \nabla \chi) : \nabla \eta \, dv_0$$

$$\text{or } = \int_{\Omega_0} T_0(\chi) : \nabla \eta \, dv_0$$

$$T_0(\chi) = \nabla \chi S(\chi) = \text{1ST PIOLA-KIRCHHOFF STRESS}$$

$$B(\chi, \eta) = \int_{\Omega_0} \partial_\theta \chi \cdot \partial_\theta \eta \, dv_0$$

$$\partial_\theta \chi \cdot \partial_\theta \eta = (\partial \chi_i / \partial \theta \chi \partial \eta_i / \partial \theta)$$

$$f(\eta) = \int_{\Omega_0} \rho_0 b_0 \cdot \eta \, dv_0$$

Figure 3

ROLLING CONTACT WITHOUT
FRICTION

\mathfrak{X} = UNILATERAL CONSTRAINT SET

$$\equiv \{ \eta \in \psi \mid \eta_2 \leq H \text{ on } \Gamma_C \}$$

VARIATIONAL INEQUALITY

FIND A MOTION $\chi \in \mathfrak{X}$ SUCH THAT

$$A(\chi, \eta - \chi) - \omega^2 B(\chi, \eta - \chi) \geq f(\eta - \chi) \quad \text{for all } \eta \in \mathfrak{X}$$

INCOMPRESSIBLE MATERIAL

$$\mathfrak{X} \equiv \{ \eta \in \psi \mid \eta_2 \leq H \text{ on } \Gamma_C, \det \nabla \eta = 1 \text{ a.e. in } \Omega_0 \}$$

Figure 4

ROLLING CONTACT WITH FRICTION

\mathcal{U} = SPACE OF VELOCITY-MOTIONS

$$\equiv \{ \eta \in \mathcal{V} \mid (\omega \partial_{\theta} \eta_1, \eta_2) \in \mathcal{V} \}$$

VIRTUAL POWER BY FRICTION

$$j : \mathcal{V} \times \mathcal{U} \rightarrow \mathcal{R}$$

$$j(\chi, \eta) \equiv \int_{\Gamma_C} \mu |T_0(\chi) n_0 \cdot n| |\omega \partial_{\theta} \eta_1 - v_0| ds_0$$

VARIATIONAL INEQUALITY \equiv B.V.P.

VARIATIONAL INEQUALITY

FIND A MOTION $\chi \in \mathcal{U} \cap \mathcal{X}$ SUCH THAT

$$A(\chi, \eta - \nabla_{\theta}^{-1} \chi) - \omega^2 B(\chi, \eta - \nabla_{\theta}^{-1} \chi) + j(\chi, \eta) -$$

$$j(\chi, \nabla_{\theta}^{-1} \chi) \geq f(\eta - \nabla_{\theta}^{-1} \chi) \quad \text{for all } \eta \in \mathcal{X}$$

$$\text{with } \nabla_{\theta}^{-1} \chi \equiv (\omega \partial_{\theta} \chi_1, \chi_2)$$

Figure 5

GENERALIZATION TO NONCLASSICAL NORMAL CONTACT/FRICTION LAWS AND REGULARIZED FRICTION FUNCTIONALS

NORMAL COMPLIANCE ON CONTACT SURFACE

$$C : \mathcal{V} \times \mathcal{V} \rightarrow \mathcal{R}$$

$$C(\chi, \eta) \equiv \int_{\Gamma_C} f(\chi_2, \omega \partial_{\theta} \chi_2) \eta_2 ds_0$$

$$f(\chi_2, \omega \partial_{\theta} \chi_2) = c_n (\chi_2 - H)_+^m n +$$

$$b_n (\chi_2 - H)_+^l n \dot{\chi}_2$$

$$\dot{\chi}_2 = \omega(\partial \chi_2 / \partial \theta)$$

VARIATIONAL INEQUALITY WITH NORMAL COMPLIANCE

FIND A MOTION $\chi \in \mathcal{U}$ SUCH THAT

$$A(\chi, \eta - \nabla_{\theta}^{-1} \chi) - \omega^2 B(\chi, \eta - \nabla_{\theta}^{-1} \chi) + C(\chi, \eta - \nabla_{\theta}^{-1} \chi)$$

$$+ j(\chi, \eta) - j(\chi, \nabla_{\theta}^{-1} \chi) \geq f(\eta - \nabla_{\theta}^{-1} \chi)$$

Figure 6

LAGRANGE MULTIPLIER METHOD

\mathcal{Q} = SPACE OF MULTIPLIER

$$\det \nabla \eta \in L^p(\Omega_0) \longrightarrow \mathcal{Q} = L^{p'}(\Omega_0)$$

$$(1/p) + (1/p') = 1$$

=====

FIND $(\chi, \rho) \in \mathcal{X} \times \mathcal{Q}$ SUCH THAT

$$A(\chi, \eta - \chi) - \omega^2 B(\chi, \eta - \chi) + I(\rho, \chi, \eta - \chi) \geq f(\eta - \chi)$$

for all $\eta \in \mathcal{X}$

$$(q, (\det \nabla \chi - 1)) = 0$$

for all $q \in \mathcal{Q}$

=====

$I(\rho, \chi, \eta)$ = TRILINEAR FORM

$$\equiv \int_{\Omega_0} \rho \operatorname{adj} \nabla \chi : \nabla \eta \, dv_0$$

$$(q, (\det \nabla \chi - 1)) \equiv \int_{\Omega_0} q (\det \nabla \chi - 1) \, dv_0$$

Figure 7

REGULARIZED FRICTION LAW

$$J_\epsilon : \mathcal{U} \times \mathcal{V} \longrightarrow \mathcal{R}$$

$$J_\epsilon(\chi, \eta) \equiv \int_{\Gamma_C} \mu |T_0(\chi) n_0 \cdot \eta| \varphi_\epsilon(|\omega \partial_\theta \chi_1 - v_0|) \eta_1 \, ds_0$$

with $\varphi_\epsilon(|w_t|) = 1$ if $|w_t| > \epsilon$
 $= |w_t|/\epsilon$ if $|w_t| \leq \epsilon$
 or $= \tanh(|w_t|/\epsilon)$

REMARK: $\lim_{\epsilon \rightarrow 0} J_\epsilon(\chi, \eta) = j(\chi, \eta) - j(\chi, \nabla_\theta \chi)$

REGULARIZED VARIATIONAL EQUALITY

FIND A MOTION $\chi_\epsilon \in \mathcal{V}$, $\rho \in \mathcal{Q}$ SUCH THAT

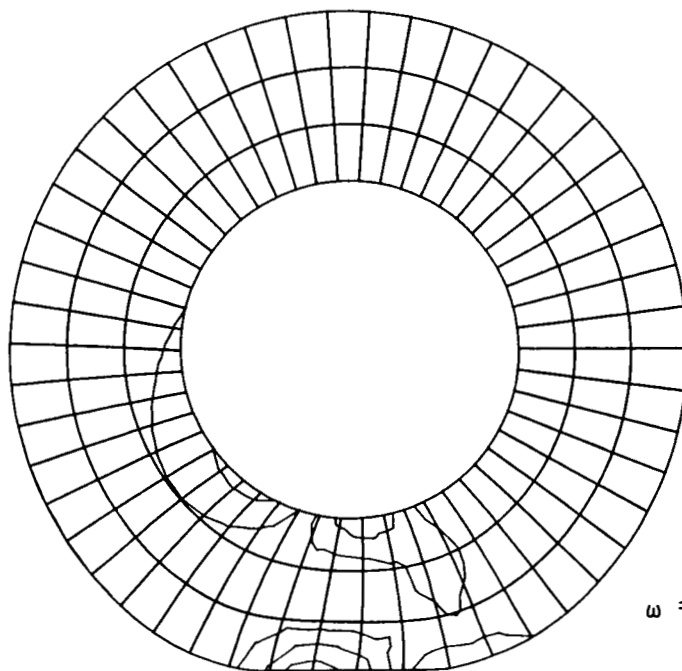
$$A(\chi_\epsilon, \eta) - \omega^2 B(\chi_\epsilon, \eta) + C(\chi_\epsilon, \eta) + J_\epsilon(\chi_\epsilon, \eta) +$$

$$I(\rho, \chi_\epsilon, \eta) = f(\eta) \quad \text{for all } \eta \in \mathcal{V}$$

$$(q, \det(\nabla \chi_\epsilon - 1)) = 0 \quad \text{for all } q \in \mathcal{Q}$$

Figure 8

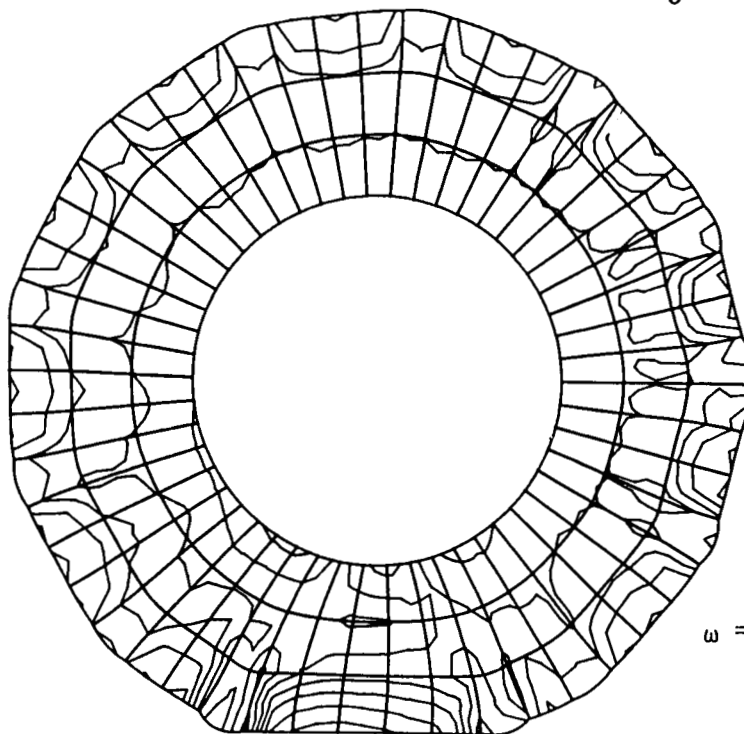
C1=80 psi C2=20 psi VIS=0 FRIC=.3 DISP=R₀ - H = .1 in.



$\omega = 300$ rpm

Figure 9

C1=80 psi C2=20 psi VIS=0 FRIC =.3 DISP=R₀ - H = .1 in.



$\omega = 300$ rpm

Figure 10

C1=80 psi C2 = 20 psi VIS=0 FRIC=.3 DISP=R₀ - H = .1 in.

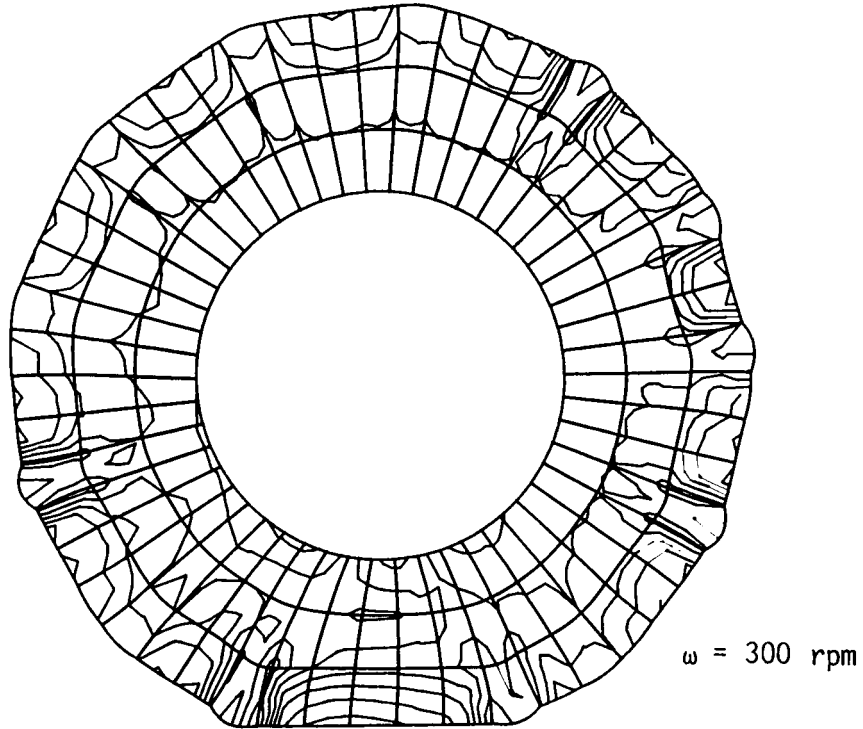


Figure 11

C1=80 psi C2=20 psi VIS=0 FRIC=.3 DISP=R₀ - H = .1 in.

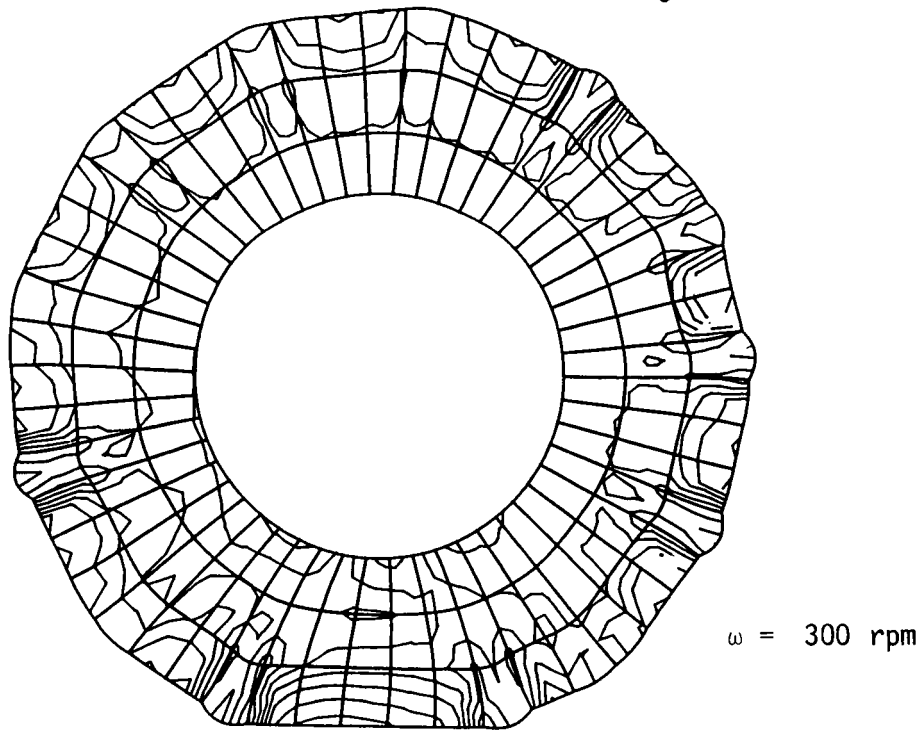


Figure 12

C1=80 psi C2=20 psi VIS=0 FRIC=.3 DISP=R₀ - H = .1 in.

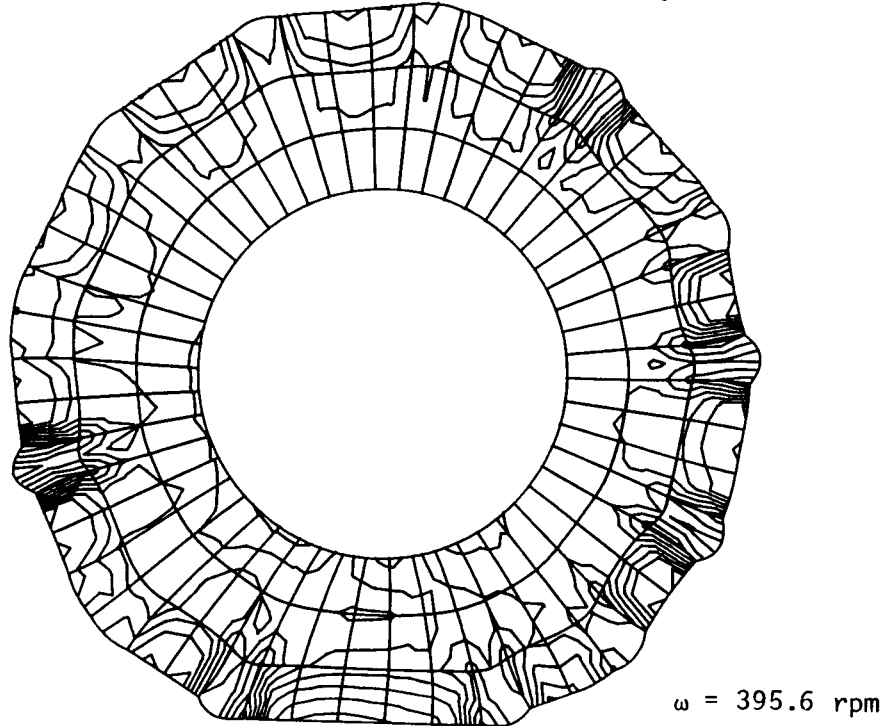
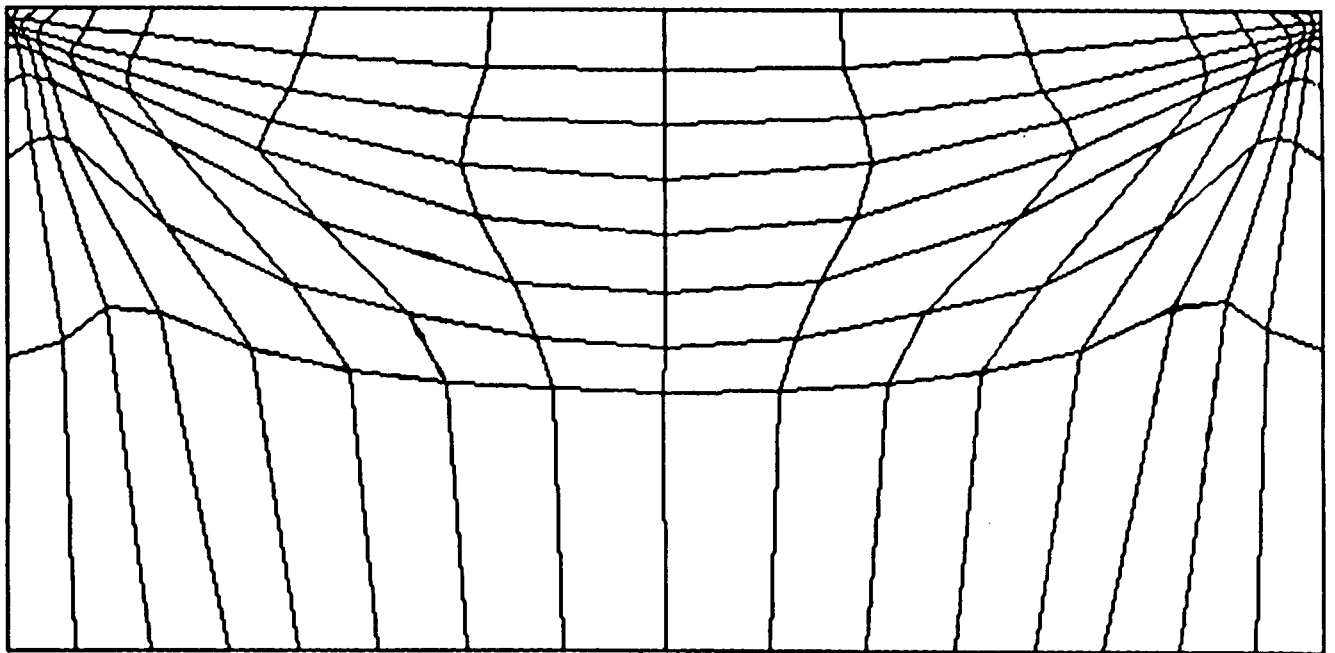


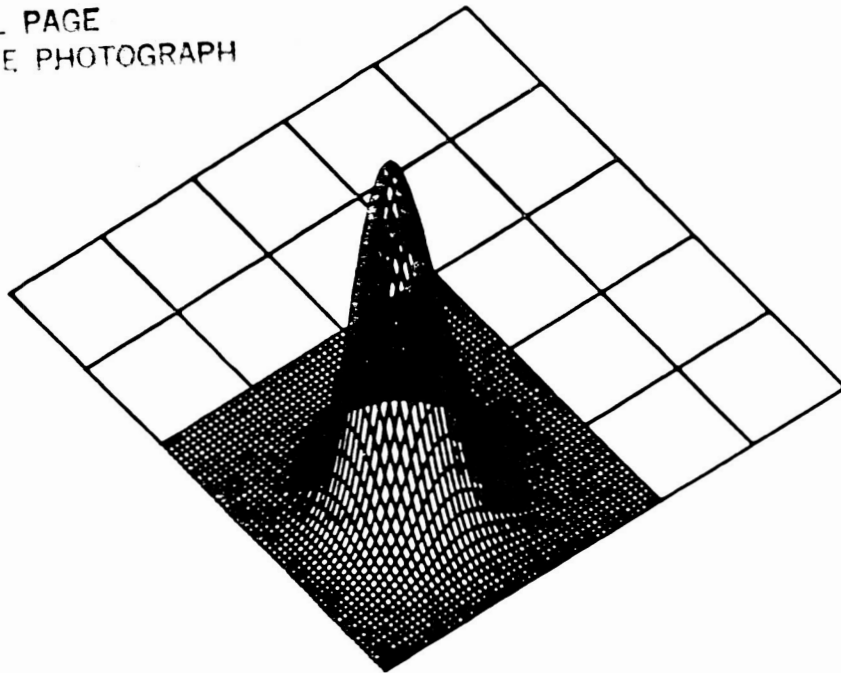
Figure 13



Driven cavity problem. Optimal mesh after 8 FE recalculations. $\alpha = 4, \beta = 4, \gamma = 0$

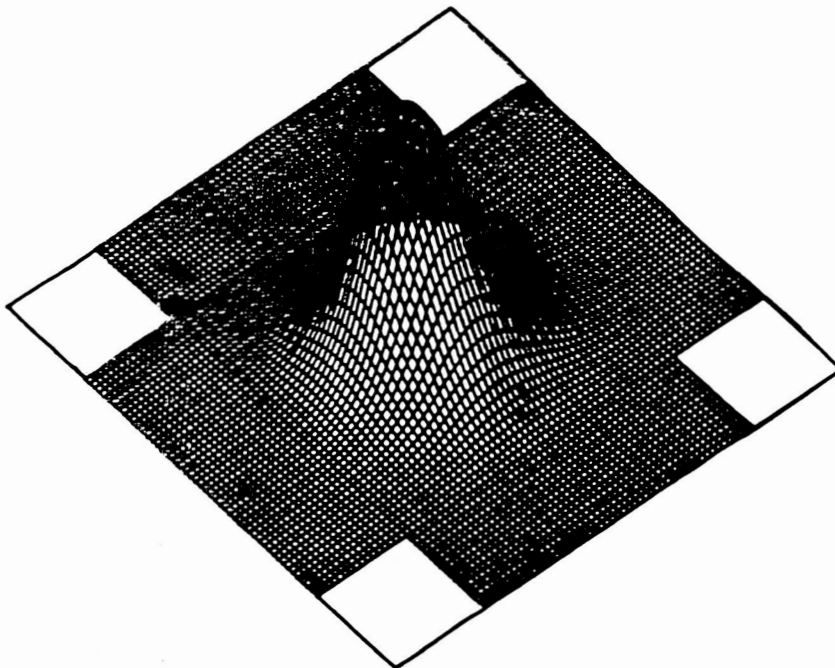
Figure 14

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



Heat equation with a dominating convection -
solution after 1 time step, $t = 0.02$

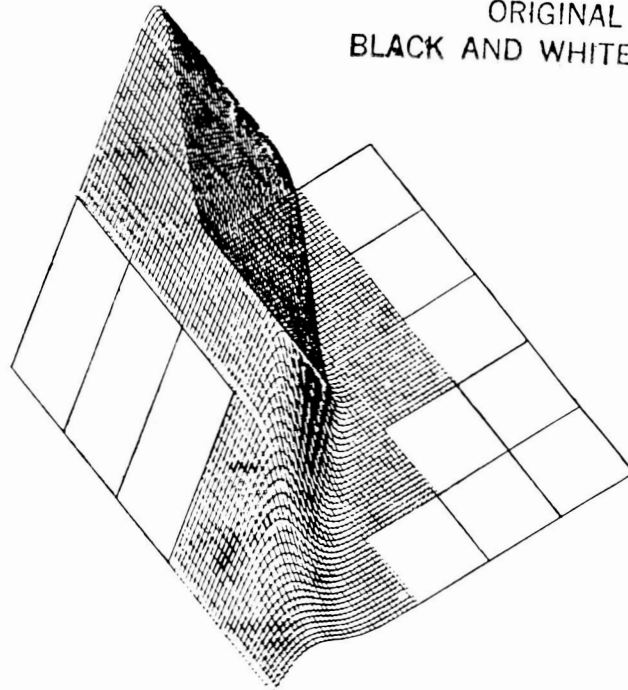
Figure 15



Heat equation with a dominating convection -
solution after 25 time steps, $t = 0.5$

Figure 16

ORIGINAL PAGE
BLACK AND WHITE PHOTOGRAPH



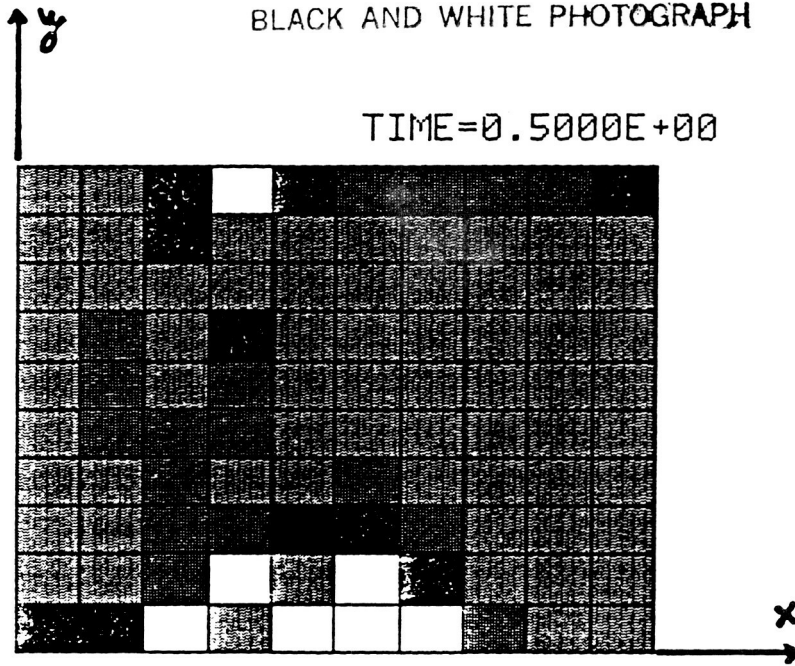
Burger's equations. First component of the solution after 1 time step, $t = 0.02$

Figure 17



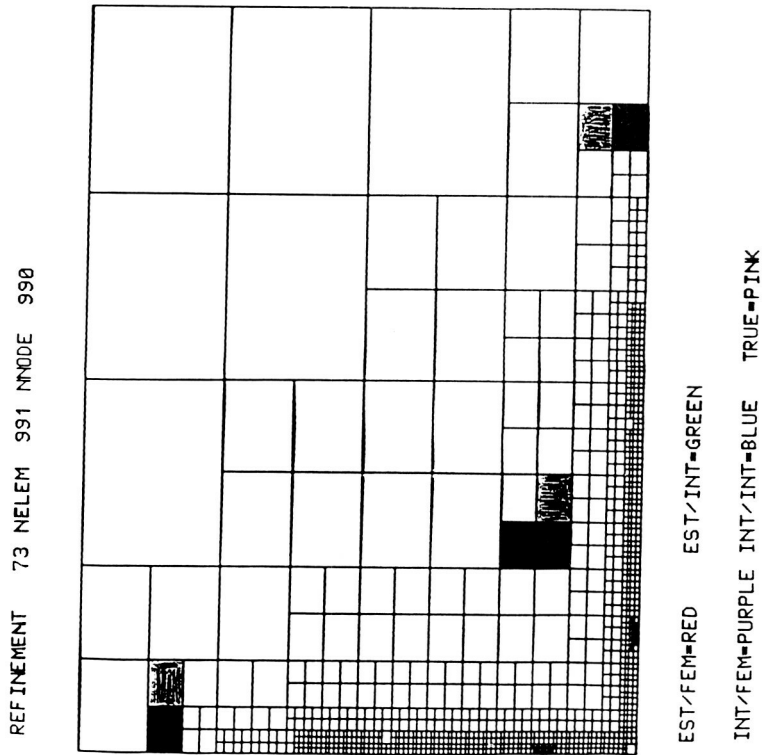
Problem with exponential solution. Deviations of the local effectivity ratios $\theta_{K,1}^0$ from unity. Parameters: $h = 2$, $\Delta t = 0.1$, $\delta = 0.25$, $M = 4$

Figure 18



Problem with exponential solution. Deviations of the local effectivity ratios $\theta_{K,5}^0$ from unity. Parameters $h = 2$, $\Delta t = 0.1$, $\delta = 0.5$, $M = 4$

Figure 19



1. Preconditioned Jacobi Conjugate Gradient Scheme
2. 4-Elements/Refinement
3. Rates = Uniform/Const. = $(C)^{1/2}$

Figure 20

CONTACT SOLUTION ALGORITHMS

John T. Tielking
Texas A and M University
College Station, Texas

EXPANDED ABSTRACT

Two algorithms for obtaining static contact solutions are described in this presentation. Although they were derived for contact problems involving specific structures (a tire and a solid rubber cylinder), they are sufficiently general to be applied to other shell-of-revolution and solid-body contact problems.

The shell-of-revolution contact algorithm is a method of obtaining a point load influence coefficient matrix for the portion of shell surface that is expected to carry a contact load. If the shell is sufficiently linear with respect to contact loading, a single influence coefficient matrix can be used to obtain a good approximation of the contact pressure distribution. Otherwise, the matrix will be updated to reflect nonlinear load-deflection behavior.

The solid-body contact algorithm utilizes a Lagrange multiplier to include the contact constraint in a potential energy functional. The solution is found by applying the principle of minimum potential energy. The Lagrange multiplier is identified as the contact load resultant for a specific deflection.

At present, only frictionless contact solutions have been obtained with these algorithms. A sliding tread element has been developed to calculate friction shear force in the contact region of the rolling shell-of-revolution tire model. This element allows a relatively general, non-Coulomb, friction law to be specified for the contact interface. It has the added advantage of allowing friction to be calculated in the continuous interface and, when coupled with the solid-body contact algorithm, will permit analytic investigation of various continuum friction theories that have been proposed.

The outline of future directions for the development of contact solution algorithms is:

- I. SHELL-OF-REVOLUTION CONTACT ALGORITHM
- II. SOLID-BODY CONTACT ALGORITHM
- III. STATIC AND ROLLING CONTACT FRICTION
- IV. FUTURE DIRECTIONS FOR CONTACT SOLUTION ALGORITHMS

PRECEDING PAGE BLANK NOT FILMED

I. SHELL-OF-REVOLUTION CONTACT ALGORITHM

A shell whose geometry and material properties are axisymmetric can be economically modeled by shell-of-revolution finite elements. The SAMMSOR/SNASOR programs (refs. 1,2), for example, permit nonlinear behavior of orthotropic shells of revolution to be calculated, including response to nonaxisymmetric loads. The algorithm described here was developed to calculate the shell deflection in response to a nodal point load, utilizing the calculated response to a sequence of harmonically varying ring loads on the node. The point load solution is then used to construct an influence coefficient matrix, from which the shell contact solution is obtained.

FINITE-ELEMENT TIRE MODEL

The tire is modeled here by an assembly of axisymmetric shell elements connected to form a meridian of arbitrary curvature and following the carcass mid-surface. The elements are homogeneous orthotropic, with moduli determined by the ply structure of a particular tire. Details of this model are given in reference 3. If the deformation is symmetric about the wheel plane only one-half of the meridian is modeled, as shown in figure 1. The finite elements are joined at nodal circles, referred to here as nodes. Node 12 in figure 1 is located at the tire bead and is given in built-in end condition.

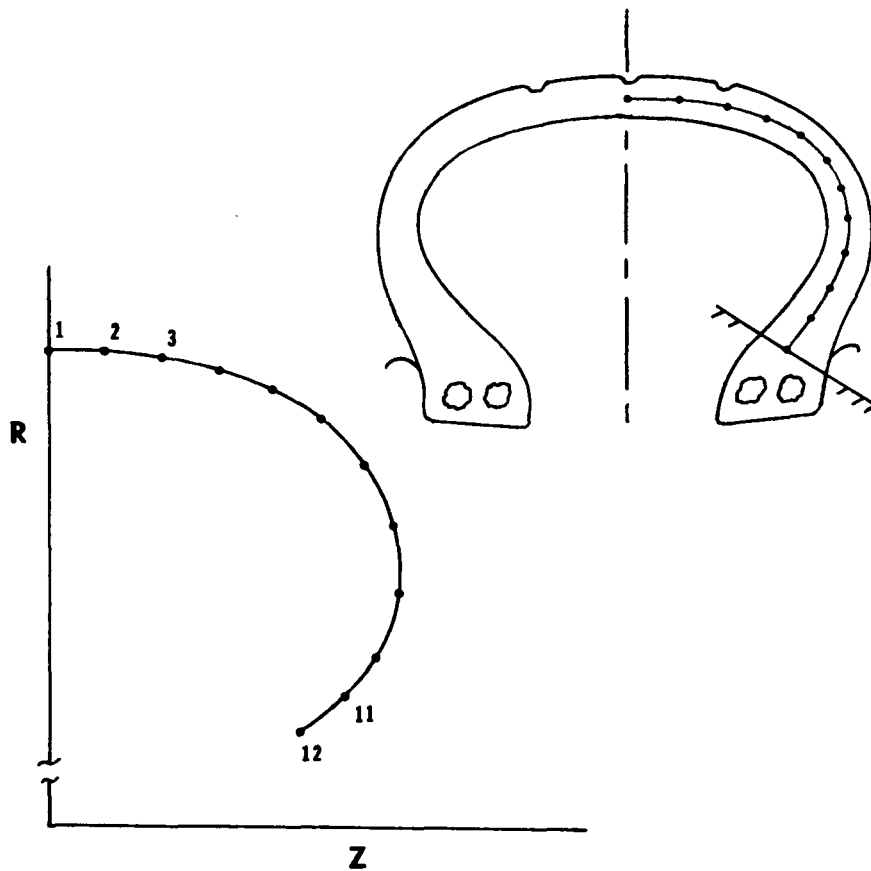
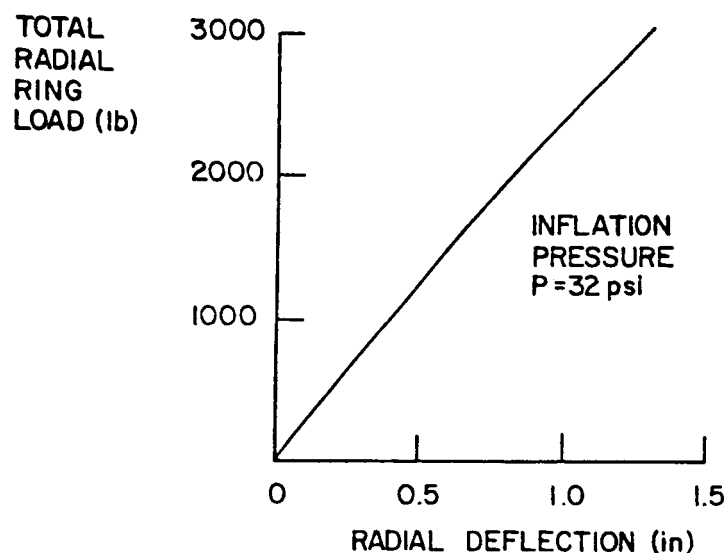


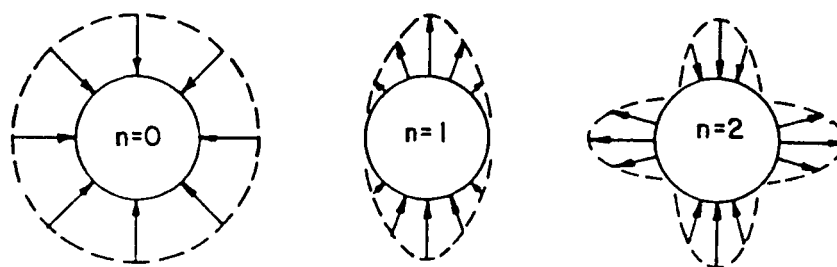
Figure 1

SINGLE HARMONIC RING LOADS

The finite-element tire model will respond to single harmonic ring loads on the nodal circles in addition to a uniform inflation pressure load. An approximately linear ring load-deflection response is obtained when an individual ring load is applied to any node of the pressurized tire model. An example ring load-deflection calculation for a passenger tire model is shown in figure 2. A harmonic sequence of stiffness matrices is obtained by applying a sequence of single harmonic ring loads to each of the nodes that may be in the tire-pavement contact region.



CROWN LOAD-DEFLECTION DATA CALCULATED WITH A UNIFORM RING LOAD APPLIED TO THE CROWN NODE



SINGLE HARMONIC RING LOADS APPLIED TO A FINITE-ELEMENT NODE

Figure 2

TRANSFER FUNCTION DEFINITION

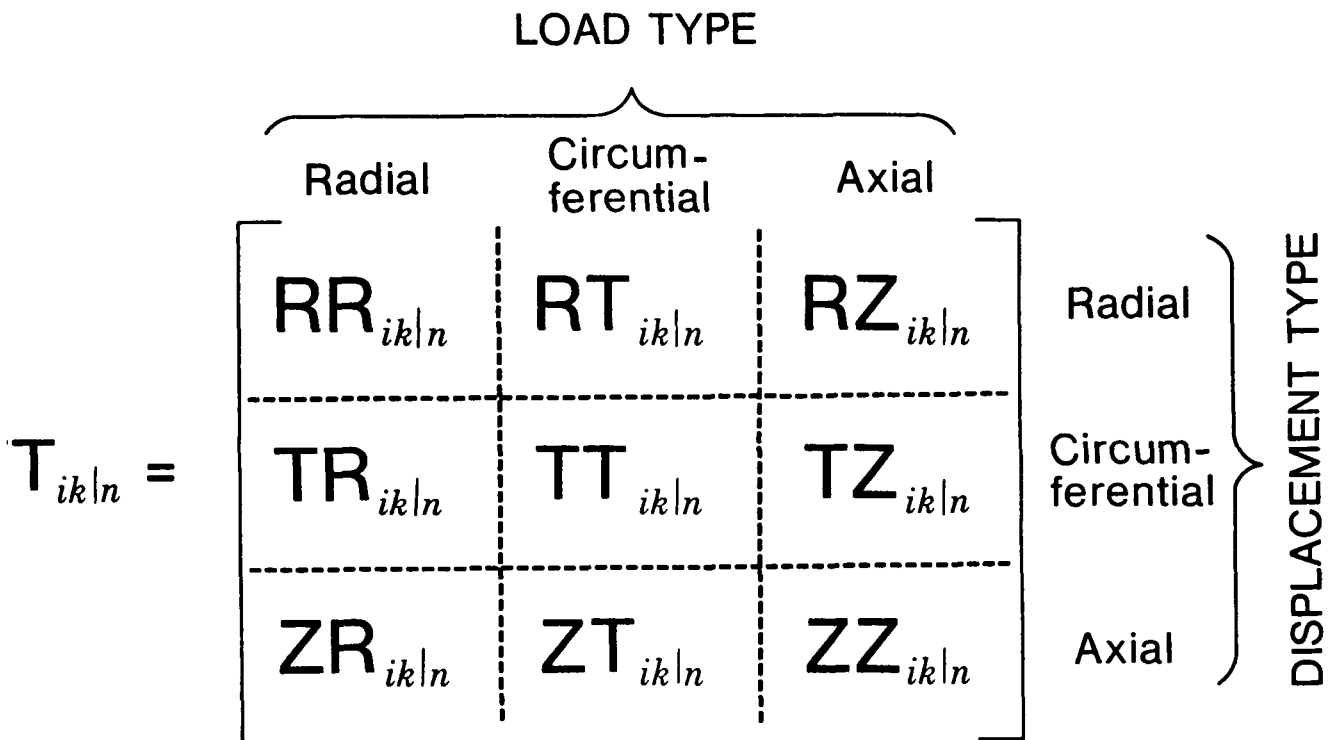
As a consequence of the linearity of the ring load-deflection response, the application of a single harmonic ring load produces a displacement field that varies circumferentially in the same harmonic as the applied ring load. The definition of the transfer function T_n as the ratio of the output and input amplitudes is given below (ref. 4). Since each node responds differently, a transfer function matrix $T_{ik|n}$ is used to store the stiffness information generated by the ring loads. The partitions of this matrix are determined by the direction of the ring load. (Fig. 3).

Single Harmonic Ring Load $A_n \cos n\theta$ (input)

Single Harmonic Displacement $B_n \cos n\theta$ (output)

$$\text{TRANSFER FUNCTION } T_n = \frac{B_n}{A_n}$$

$T_{ik|n} = n^{\text{th}}$ harmonic transfer function relating displacement of node i to an n^{th} harmonic ring load on node k



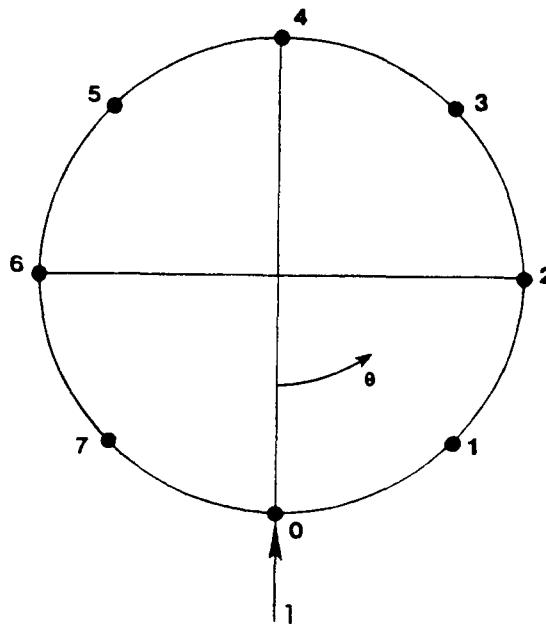
$$n=0,1, \dots, N/2$$

Figure 3

POINT LOAD VECTOR {p} AND THE DISCRETE FOURIER TRANSFORM (DFT)

This application of the discrete Fourier transform uses an even number of points (N), equally spaced around the circumference. The example shown in figure 4 uses N=8 points. A unit load is applied at any point, say point 0. The DFT of the load vector yields a set of N coefficients, G_j , which are approximate values of the coefficients of the conventional Fourier series defined on the continuous interval $0 \leq \theta \leq 2\pi$ and representing the unit point load. The point load is applied, sequentially, in the radial, axial, and circumferential directions.

INFLUENCE COEFFICIENT GENERATION



$$\{p\} = \{1, 0, 0, 0, 0, 0, 0, 0\} \quad \text{load vector}$$

$$\text{DFT } G_j = \frac{1}{N} \sum_{k=0}^{N-1} g_k w_e^{jk} \quad w_e = e^{-i2\pi/N}$$

$$g_k = \{p\}, \quad G_j = \frac{1}{N} \quad j = 0, 1, \dots, N-1$$

Figure 4

INVERSE DISCRETE FOURIER TRANSFORM (IDFT) AND THE INFLUENCE COEFFICIENTS

Having the unit point load represented by a conventional Fourier series, whose coefficients a_n are approximately given by the DFT coefficients, the transfer functions $T_{ik|n}$ are applied, on each harmonic, to obtain the coefficients b_n of the Fourier series representing the response of the nodal circle to the unit point load. The inverse discrete Fourier transform is then used to evaluate the displacements, u_m , at the N points. These displacements are the elements of the influence coefficient matrix $[A_{ijkl}]$. (Fig. 5).

$$\text{INPUT SERIES COEFFICIENTS } a_n \approx G_n = \frac{1}{N}$$

$$\text{OUTPUT SERIES COEFFICIENTS } b_n = a_n T_{ik|n} = \frac{1}{N} T_{ik|n}$$

$$\text{DFT OF DISPLACEMENT VECTOR } G_n \approx b_n$$

$$\text{IDFT } u_m^{ik} = \sum_{n=0}^{N-1} G_n W_n^{-mn} \quad m = 0, 1, \dots, N-1$$

$$\text{INFLUENCE COEFFICIENTS } A_{ijk\ell} = u_{j-1}^{ik} \quad j = 1, 2, \dots, N$$

$$\text{SHIFT: } A_{ijkl} = u_{j-\ell}^{ik} \quad j = \ell, \ell+1, \dots, N$$

$$\text{SYMMETRY: } A_{klij} = A_{ijkl}$$

$$\{d_{ij}\} = [A_{ijkl}]\{P_{kl}\}$$

Figure 5

TOROIDAL SHELL CONTACT SCHEMATIC

A cylindrical coordinate system is used to locate points on the toroidal surface. The coordinates r , θ , and z indicate the radial, circumferential, and axial directions, respectively. The tire equator lies in the r - θ plane (wheel plane) and a tire meridian is in an r - z plane.

After the inflation solution has been obtained, the tire model is deflected against a frictionless, flat surface. The contacting surface is perpendicular to the wheel plane and positioned at the specified loaded radius R_l , as shown in figure 7. The vertical load and the contact pressure distribution are unknown, *a priori*.

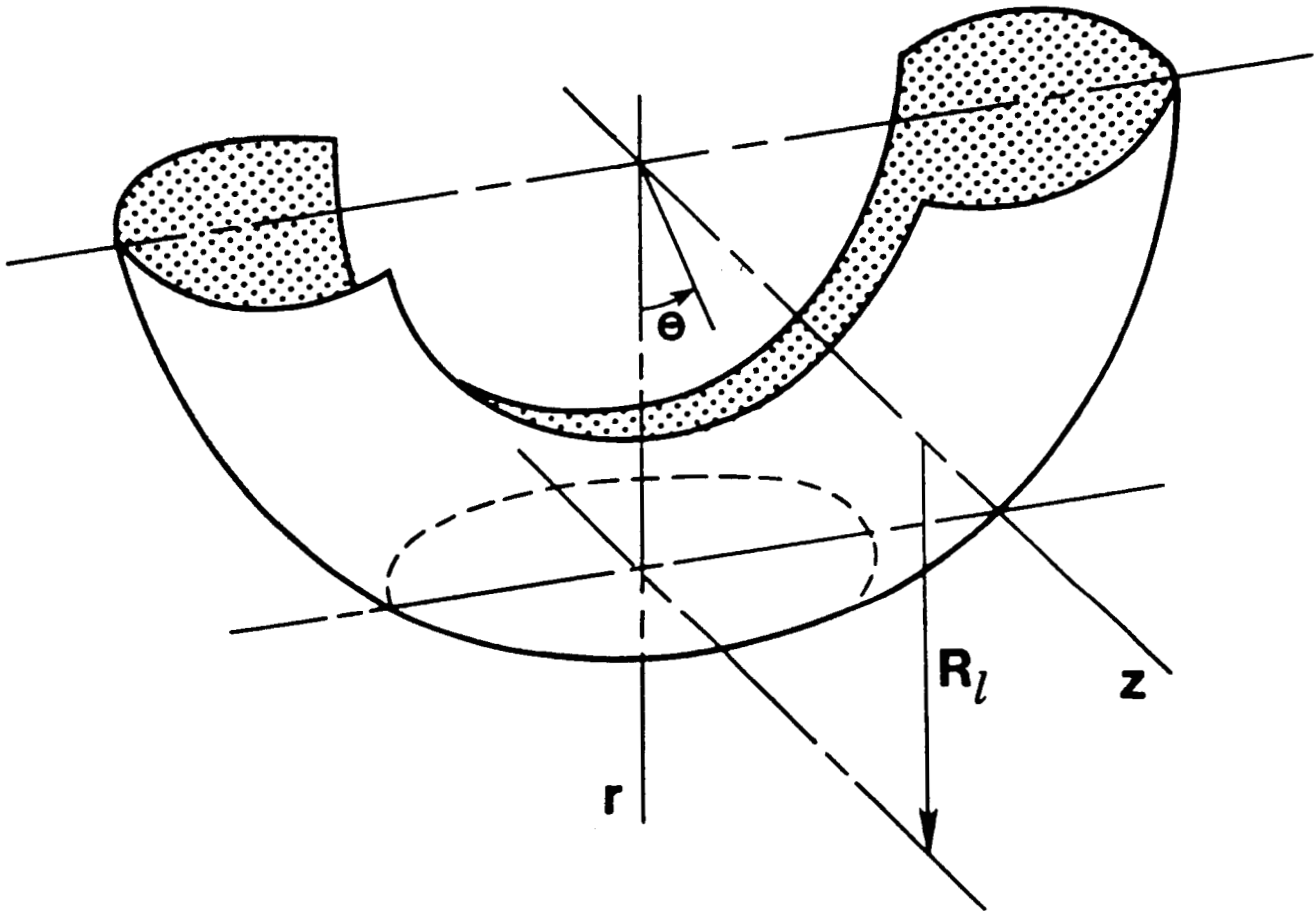


Figure 7

DEFLECTED MERIDIAN

The deflected shape of the meridian passing through the center of contact is shown in figure 8. This shape is calculated by the finite-element tire model for the specified tire deflection of one inch. The tire load that will produce a one inch tire deflection is calculated to be 10,590 lb. Figure 8 also shows the meridian prior to inflation and the calculated shape of the meridian of the inflated tire, prior to contact loading. These finite-element meridians follow the carcass midsurface, as indicated in figure 1. Geometric and material property data on the Space Shuttle nose gear tire were used for the calculated results shown in figures 8, 9, and 10.

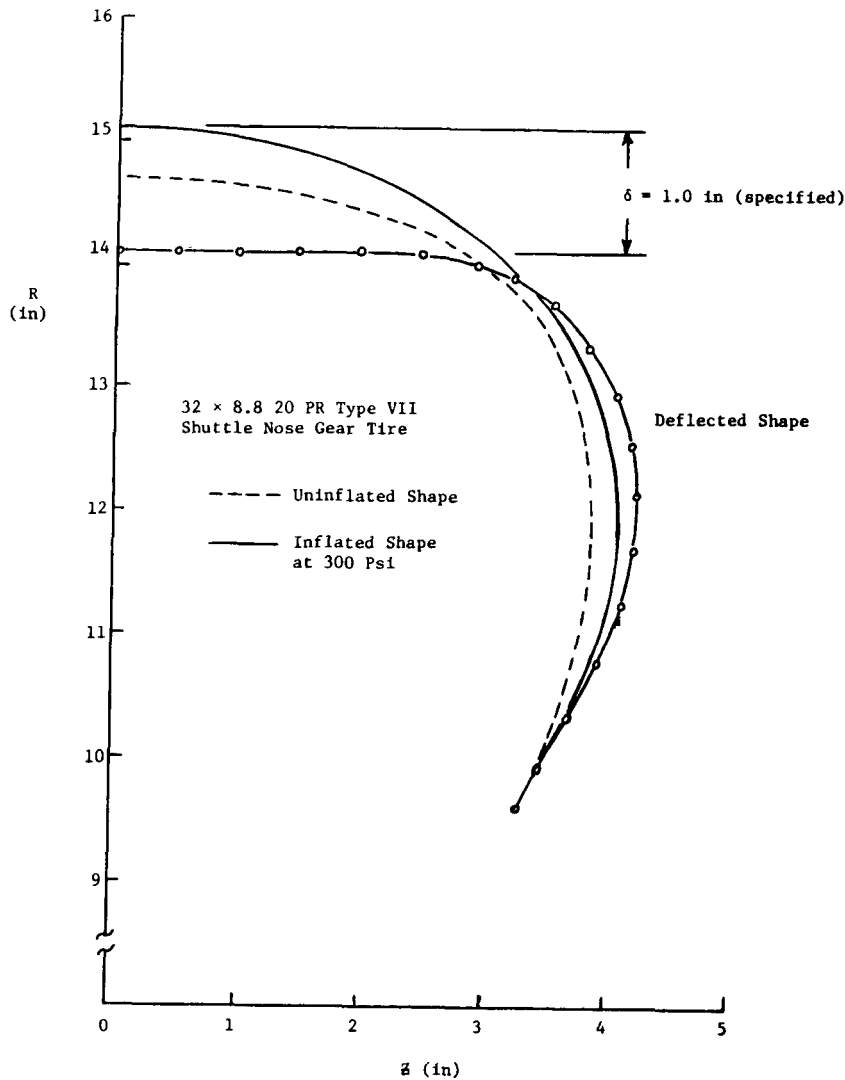


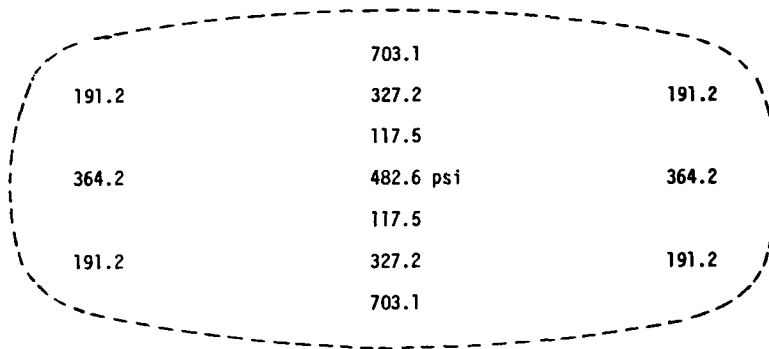
Figure 8

CONTACT PRESSURE DISTRIBUTIONS

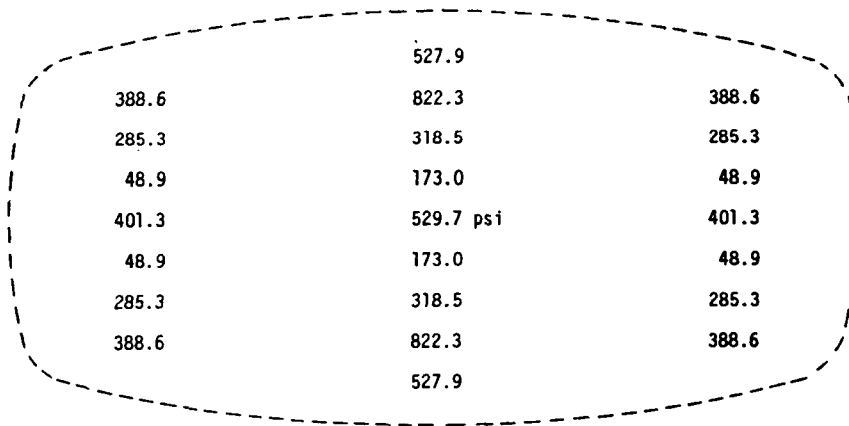
The static contact pressure values (psi) calculated for two different loads on the Shuttle nose gear tire are shown in figure 9. The number of finite-element points in the contact region increases as the tire load increases. A rough estimate of the contact boundary is obtained by extrapolation of the pressure distribution. Integration of the pressure distribution gives the tire load.

NOSE GEAR TIRE CONTACT PRESSURE DISTRIBUTIONS

32x8.8 20 PR TYPE VII
Inflation Pressure = 300 psi



(a) $\delta = 0.75$ in, $F_z = 5,708$ lb



(b) $\delta = 1.00$ in, $F_z = 10,590$ lb

Figure 9

TIRE LOAD VERSUS TIRE DEFLECTION

An important test of a tire model is its ability to calculate a static load-deflection curve. Figure 10 compares the load-deflection curve calculated for the Shuttle nose gear tire with measured data for a similar aircraft tire. Although these are both 32 x 8.8 Type VII tires, constructional details can alter the load-deflection curve (and many other aspects of tire behavior). The cord used in the test tire is unknown and may be quite different from the nylon cord in the Space Shuttle tire.

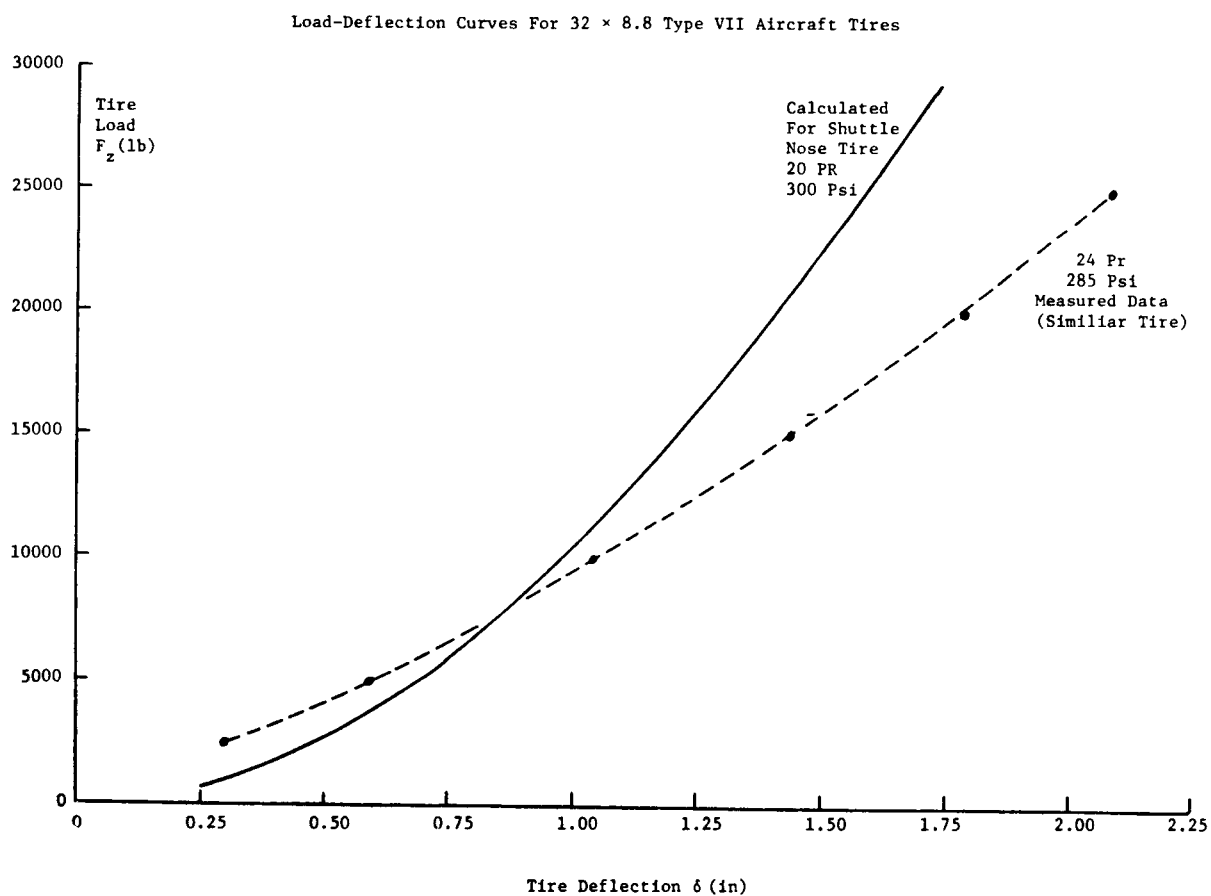


Figure 10

II. SOLID-BODY CONTACT ALGORITHM

A solid body with a cylindrical surface is often loaded in contact against a rigid surface. The contact load may revolve around the body, as in the case of a roller or a solid tire, or may remain stationary if the cylinder is used as a support cushion.

Interfacial friction, present in all contact problems, is currently an active field of research. The study of frictional behavior is facilitated if the contact region is relatively large, as is produced when the body is highly deformable. This makes it easier to calculate distributions of normal pressure and tangential motion (slip) in the interface. In the case of rubber contact, the behavior deviates sufficiently from the Coulomb friction law that other, more physically realistic laws, can be easily tested. Since friction is a microscopic phenomenon, a contact solution giving continuous distributions of interfacial pressure and slip is desirable for analytic purposes. The contact algorithm described here provides a continuum solution for frictionless contact, the first step toward analysis of friction in the continuous contact interface.

A PLANE STRAIN CONTACT PROBLEM

An elastic semicylinder of radius R is bonded to a fixed surface, figure 11(a). A contact load is applied by a rigid plate that deflects the semicylinder as shown in figure 11(b).

The problem is formulated in terms of cylindrical material coordinates (r, θ, z) which identify points in the undeformed body, B_0 . A point P_0 in B_0 is located by Cartesian coordinates x_1 and x_2 axes shown in figure 11(a) and

$$x_1 = r \cos \theta \qquad x_2 = r \sin \theta \qquad x_3 = z$$

The contact load is assumed to produce a plane strain deformation. Point P_0 moves to position P in the deformed body, B . Point P is located by the Cartesian coordinates y_1 . With plane strain, $y_1 = y_1(r, \theta)$, $y_2 = y_2(r, \theta)$, and $y_3 = \lambda_3 z$ where $\lambda_3 = 1$ is a specified constant extension ratio.

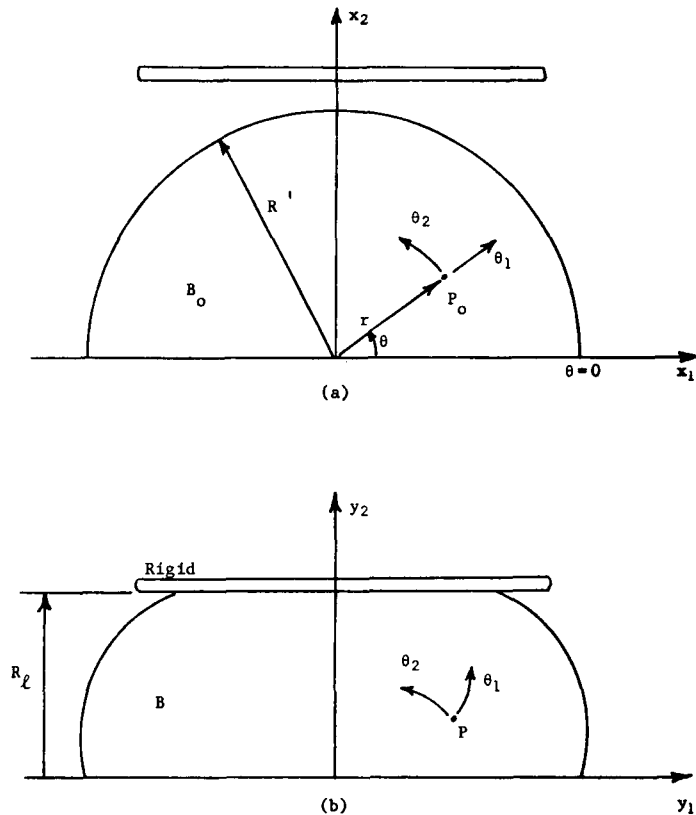


Figure 11

GEOMETRIC DESCRIPTION

The metric tensors g_{ij} and G_{ij} , given below, completely describe the elastic semicylinder before and after deformation. Since y_3 is known *a priori*, the problem is solved by finding the functions $y_1(r, \theta)$ and $y_2(r, \theta)$ which determine G_{ij} . The displacement field is not utilized in this formulation but it can, of course, be found when x_i and y_i are known. (Fig. 12).

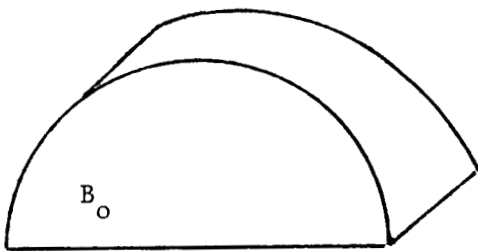
$$g_{ij} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & r^2 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

$$g^{ij} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & r^{-2} & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

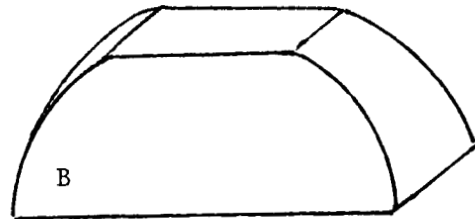
$$G_{ij} = \begin{bmatrix} y_{\alpha,1} y_{\alpha,1} & y_{\alpha,1} y_{\alpha,2} & 0 \\ y_{\alpha,2} y_{\alpha,1} & y_{\alpha,2} y_{\alpha,2} & 0 \\ 0 & 0 & \lambda_3^2 \end{bmatrix} \quad \begin{array}{l} \text{(implied sum} \\ \text{with } \alpha = 1,2) \end{array}$$

The Green/Saint-Venant strain tensor components are defined as

$$\gamma_{ij} = \frac{1}{2}(G_{ij} - g_{ij})$$



Undeformed
Metric $[g_{ij}]$



Deformed
Metric $[G_{ij}]$

Figure 12

MATERIAL DESCRIPTION

The material is assumed to be hyperelastic so that its constitutive properties are contained in a strain energy density, W . Isotropy is also assumed. For plane strain of an isotropic material, the strain energy is known to be a function of only the first and third strain invariants, I_1 and I_3 (ref. 5)

$$W = W(I_1, I_3)$$

For general deformation, the strain invariants are given by

$$I_1 = g^{ij} G_{ij} \qquad I_3 = G/g$$

where $g = \det [g_{ij}] = r^2$ for the semicylinder and $G = \det [G_{ij}]$.

When the material is also assumed to be incompressible ($I_3 = 1$) the constitutive behavior is not completely determined by the strain energy density. Hydrostatic pressure becomes an additional unknown, which can be determined as a Lagrange multiplier (ref. 6). This difficulty is avoided if a compressible material model is used.

The material description selected for the contact problem solved here is the compressible neo-Hookean model developed for continuum rubber by Blatz and Ko (ref. 7). The Blatz-Ko model may be expressed as

$$W(I_1, I_3) = \frac{1}{2} \mu (I_1 - 3I_3) + \frac{1}{k} K \left(\frac{1}{3} - \frac{k}{k-1} + \frac{I_3^{1/2} (1-k)}{k-1} \right)$$

where μ is the classical shear modulus, K is the bulk modulus, and k is a parameter related to atomic repulsion. When $I_3 = 1$, the Blatz-Ko model reduces to the neo-Hookean model for incompressible material. For small strains it reduces to the energy density giving Hooke's law for compressible isotropic material.

CONTACT CONSTRAINT

The assumption of frictionless contact with a rigid surface, positioned perpendicular to the y_2 axis (see fig. 11(b)), implies a geometric constraint only on the solution function $y_2(r, \theta)$. Within the contact region, whose extent is not known *a priori*, the deformed surface is flat and it is known that $y_2 = R_\ell$ where R_ℓ is the specified location of the contact surface. Outside the contact region, and in the interior of B, the solution must satisfy $y_2 < R_\ell$. This inequality constraint on y_2 is converted to an equality constraint by introducing a new function $s(r, \theta)$, defined by the following equation

$$y_2 + s^2 = R_\ell \quad (\text{constraint equation})$$

which is valid everywhere on the boundary and in the interior of B. The function $s(r, \theta)$, called a slack variable, has been used previously in optimization problems with an inequality constraint. Reference 8 gives several examples of the use of slack variables.

The contact problem is solved by minimizing the strain energy in B, subject to the constraint equation given above. Since plane strain is assumed, the energy is uniform along the axis of the semicylinder. Using symmetry, integration of the energy density is taken over one-half of the r - θ plane contained in B_0 .

The constraint equation is brought into the energy density functional by means of a Lagrange multiplier function $\lambda(r, \theta)$. The contact problem is then governed by the following functional

$$I(y_1, y_2, s, \lambda) = \iint F(r, \theta, y_1, y_2, s, \lambda) \, drd\theta$$

where

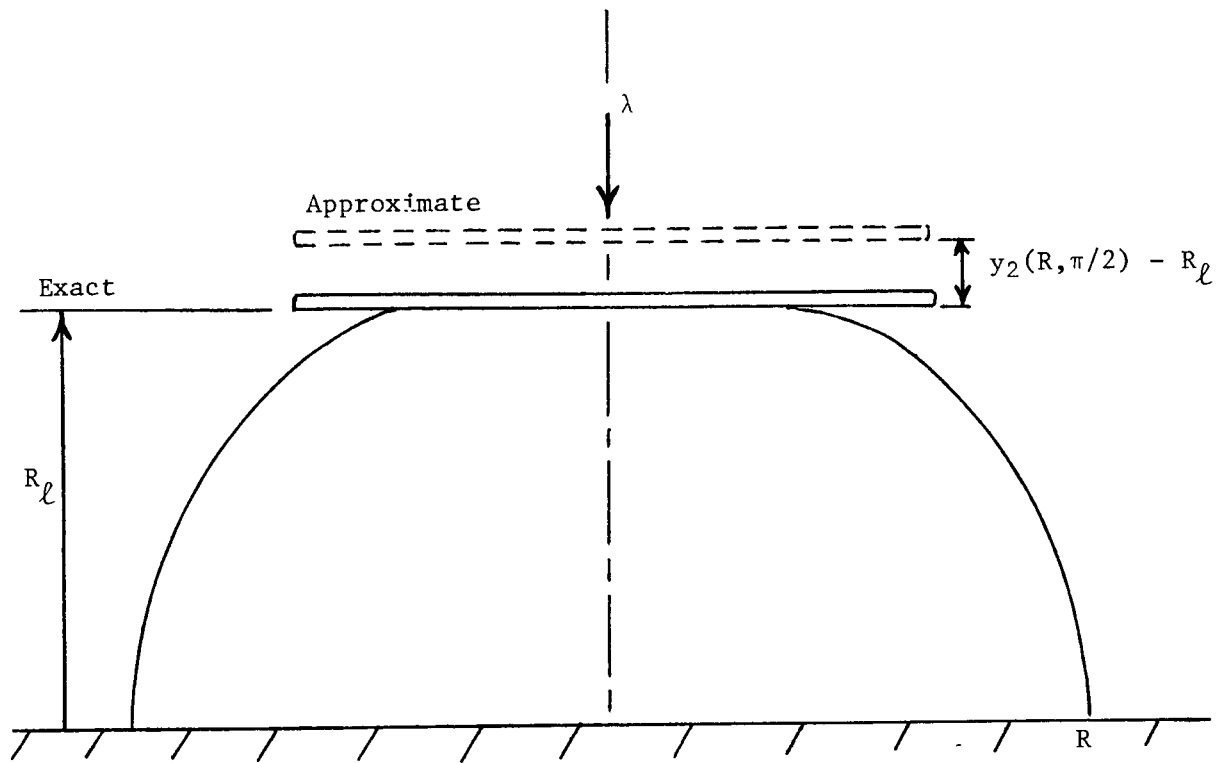
$$F = rW(y_{\alpha, \beta}) + \lambda (y_2 + s^2 - R_\ell)$$

Although rW is positive definite, F is not positive definite due to the addition of the constraint. Therefore, I may only be regarded as being made stationary instead of minimized by equilibrium solution functions. The integral of W , however, is minimized by the equilibrium solution and this is used as a check during the solution finding process.

Through additional analysis (J. T. Tielking, Texas A and M University, unpublished data) the Lagrange multiplier function is shown to be an unknown constant, identified as the resultant load in the contact region. The constraint condition may then be removed from the integral, the slack variable is no longer needed, and the contact problem is now governed by

$$I(y_\alpha, \lambda) = \iint rW(y_{\alpha, \beta}) \, drd\theta + \lambda (y_2(R, \pi/2) - R_\ell)$$

The solution is obtained by finding $y_\alpha(r, \theta)$ and the constant λ which make $I(y_\alpha, \lambda)$ stationary. (Fig. 13).



Example: $M = N = 2$

$$y_2(R, \pi/2) = R + (b_{11} - b_{12})R + (b_{21} - b_{22})R^2$$

Figure 13

SOLUTION FUNCTIONS

The numerical solution is obtained by application of the principle of stationary potential energy (ref. 6), using the functional $I(y_\alpha, \lambda)$ in which λ is an unknown constant. The solution functions y_α are taken as two-dimensional finite series

$$y_1(r, \theta) = x_1(r, \theta) + \sum_{i=1}^M \sum_{j=1}^N a_{ij} y_1^{ij}(r, \theta)$$

$$y_2(r, \theta) = x_2(r, \theta) + \sum_{i=1}^M \sum_{j=1}^N b_{ij} y_2^{ij}(r, \theta)$$

where a_{ij} and b_{ij} are unknown coefficients. The symmetry and geometric boundary conditions evident in figure 11 are met by

$$y_1^{ij}(r, \theta) = r^i \sin(2j\theta)$$

$$y_2^{ij}(r, \theta) = r^i \sin[(2j-1)\theta]$$

The above functions allow the energy density to be integrated, thereby reducing the functional I to an algebraic function of the $2 \times M \times N + 1$ unknown constants a_{ij} , b_{ij} , and λ

$$I = I(a_{ij}; b_{ij}; \lambda)$$

The functional I is made stationary by the constants obtained from the following set of simultaneous nonlinear equations:

$$\frac{\partial I}{\partial a_{ij}} = 0 \quad \text{and} \quad \frac{\partial I}{\partial b_{ij}} = 0 \quad \text{for } i = 1, 2, \dots, M \quad \text{and } j = 1, 2, \dots, N$$

$$\frac{\partial I}{\partial \lambda} = y_2(R, \pi/2) - R_\ell = 0$$

This system is solved in an iterative manner by the Newton-Raphson method. Using the starting values $a_{ij} = b_{ij} = \lambda = 0$, five or six iterations (which give successive corrections to these constants) are usually sufficient. The iterations are continued until the corrections appear to have negligible effect on the solution functions $y_\alpha(r, \theta)$. The energy density is evaluated after each iteration to check for minimization.

DEFORMATION SOLUTION

Numerical results have been obtained using material constants $\mu = 75$ psi (shear modulus), $K = 475,000$ psi (bulk modulus), and $k = 13.3$ in the Blatz-Ko model. The values of K and k are taken from reference 7 where they are shown to give a good fit to hydrostatic compression data on Butyl tread rubber (polyisobutylene). The shear modulus is believed to be a realistic estimate, based on Treloar's statement (ref. 9) that the shear modulus of rubber is lower than the bulk modulus by a factor of about 10^4 .

The computer-generated drawing below shows coordinate circles and radii before deformation (x_1 and x_2 , dashed lines) and the deformed configuration (y_1 and y_2 , solid lines) of these circles and radii. The deformation is produced by a 10-percent deflection of the contacting surface (shown dashed). The deformation solution, $y_1(r, \theta)$ and $y_2(r, \theta)$, is obtained in a 16-term series for each function; the strain energy is minimized by the coefficients a_{ij} and b_{ij} for $i=1,2,3,4$ and $j=1,2,3,4$, found after six Newton-Raphson iterations. This computation took 30 seconds of CPU time on a mainframe computer (Amdahl 470/V8). The Lagrange multiplier obtained in this solution is $\lambda=93.1$, interpreted as a 93.1 lb load needed for a 10-percent deflection if the semicylinder extends 1 inch in the z -direction. (Fig. 14).

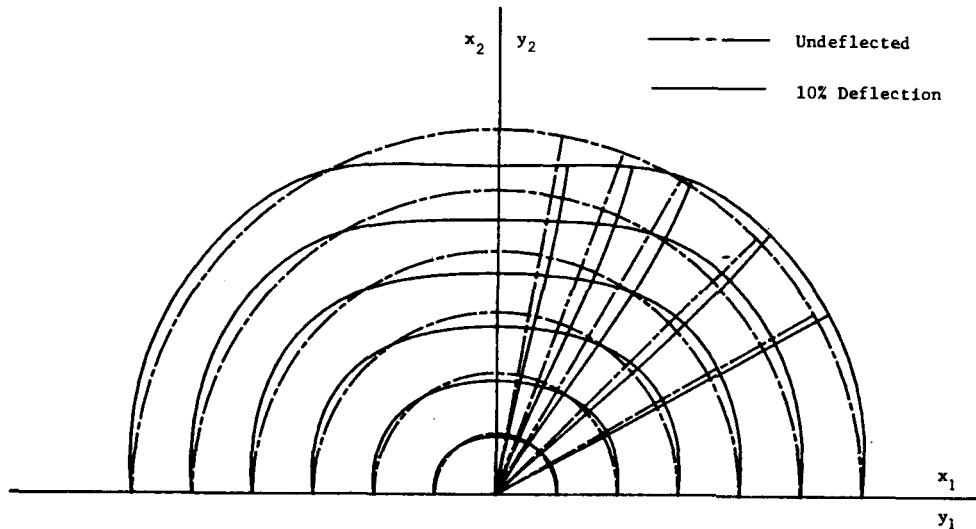


Figure 14

III. STATIC AND ROLLING CONTACT FRICTION

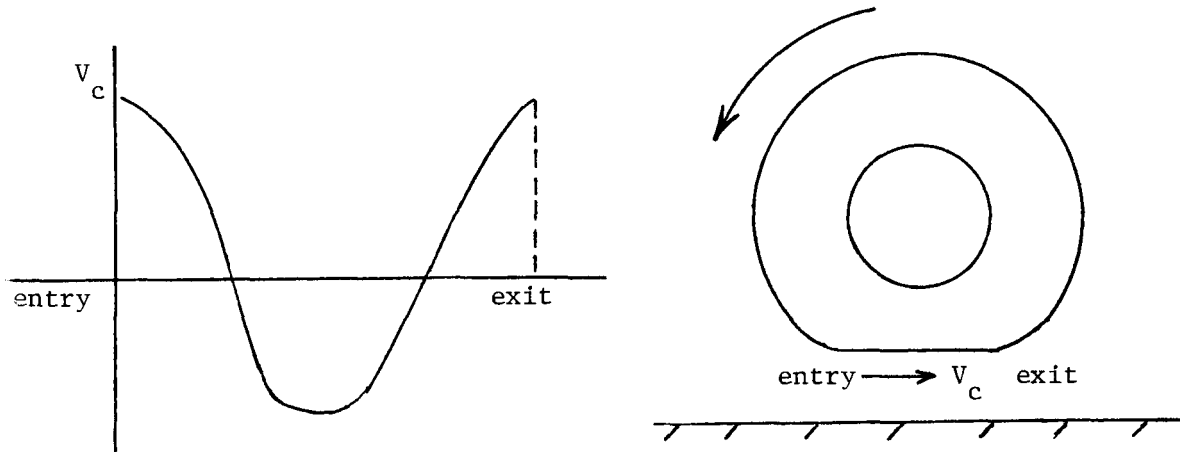
Static Contact Friction. A body is brought into static contact by motion perpendicular to the contact plane. During this motion, the contact boundary expands until the resultant of the normal contact pressure reaches equilibrium with the external load applied to the body. As the contact region is formed, shear forces are generated by tangential motion of contacting surface points. These shear forces are frictional and transient, reaching equilibrium levels when the body itself comes into equilibrium. Although the body may be assumed elastic, and thus conservative, the frictional shear forces are not conservative.

The formidable problem of calculating a static contact solution including the effect of friction is alleviated somewhat by assuming Coulomb's law of friction is valid in the contact region. An algorithm for including Coulomb friction in a static contact problem has been developed by Rothert et al. (ref. 10). Although Coulomb friction may be taken for an approximate analysis of frictional contact, mathematical and physical uncertainties arise when it is assumed. Nonclassical friction laws have been proposed by Oden and his coworkers (e.g., ref. 11). Although these appear to have been developed mainly for metals, they may be applicable to more deformable material such as rubber.

Rolling Contact Friction. This discussion is limited to steady rolling under a constant load. Neglecting hysteretic effects in the body, the power input to maintain steady rolling is balanced by the work rate of the friction forces in the contact region. In steady rolling, a contact region, whose boundary is fixed by the load, is continuously generated. The normal pressure and sliding velocity at a given location within the contact boundary do not change with time so steady-state frictional behavior is maintained. The rolling contact problem with friction is therefore much easier to analyze and provides a mechanism for the study of nonclassical friction theories. An algorithm for calculating friction in the contact region of a tire rolling at constant velocity will be described next.

SLIDING TREAD MODEL

This is a tread element model developed to convert a frictionless rolling contact solution into a solution for rolling contact with friction. The element passes through the contact region with the velocity found for frictionless contact. This is termed the carcass velocity, V_c , whose distribution is symmetric about the center of the footprint as sketched below



For a free-rolling tire, the amplitude of the footprint sliding velocity is very small. At 60 mph (1056 ips), the peak V_c is calculated (by the author) to be about 50 ips in a frictionless footprint.

In free-rolling, the normal contact pressure distribution, p , is essentially unchanged by friction. The sliding velocity distribution, however, is significantly altered in an interactive manner. A hysteretic theory of tire-pavement friction proposed by Schapery (ref. 12) gives the dependence of the friction force, F_s , on the actual sliding velocity, V_s , and normal pressure, p , at a point in the contact region. This is expressed as

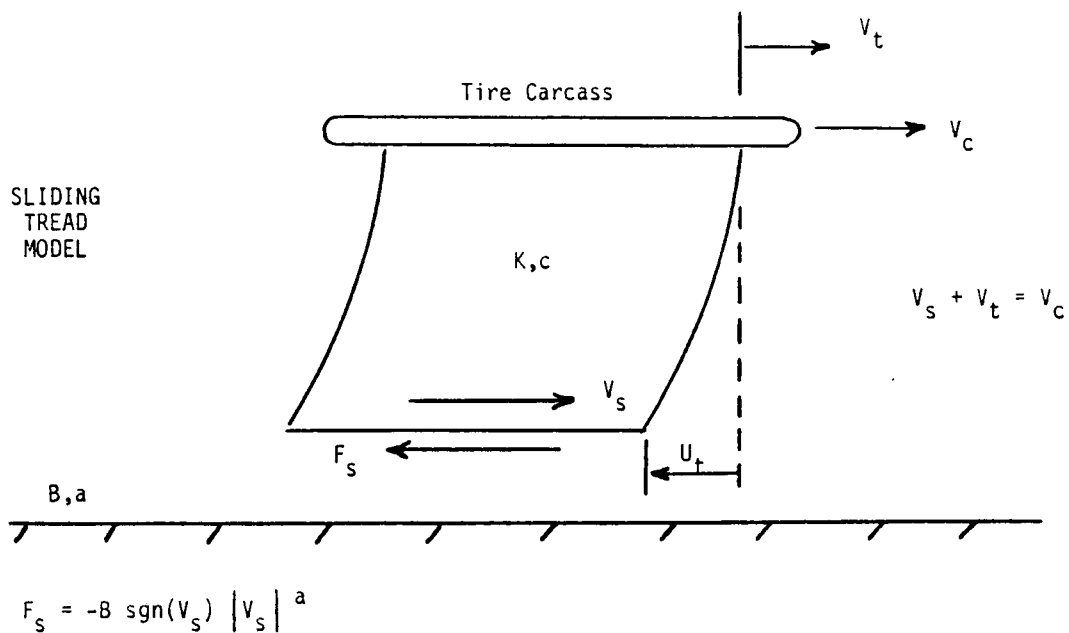
$$F_s = -B \operatorname{sgn}(V_s) |V_s|^a \times p^b \quad (\text{hysteretic theory})$$

where a , b , and B are material friction properties. The sliding tread element model, shown in figure 15(a), is viscoelastic with stiffness and damping parameters K and c . Sliding friction, F_s , causes the element to deform, thereby influencing the sliding velocity V_s . The following nonlinear differential equation is integrated to calculate V_s .

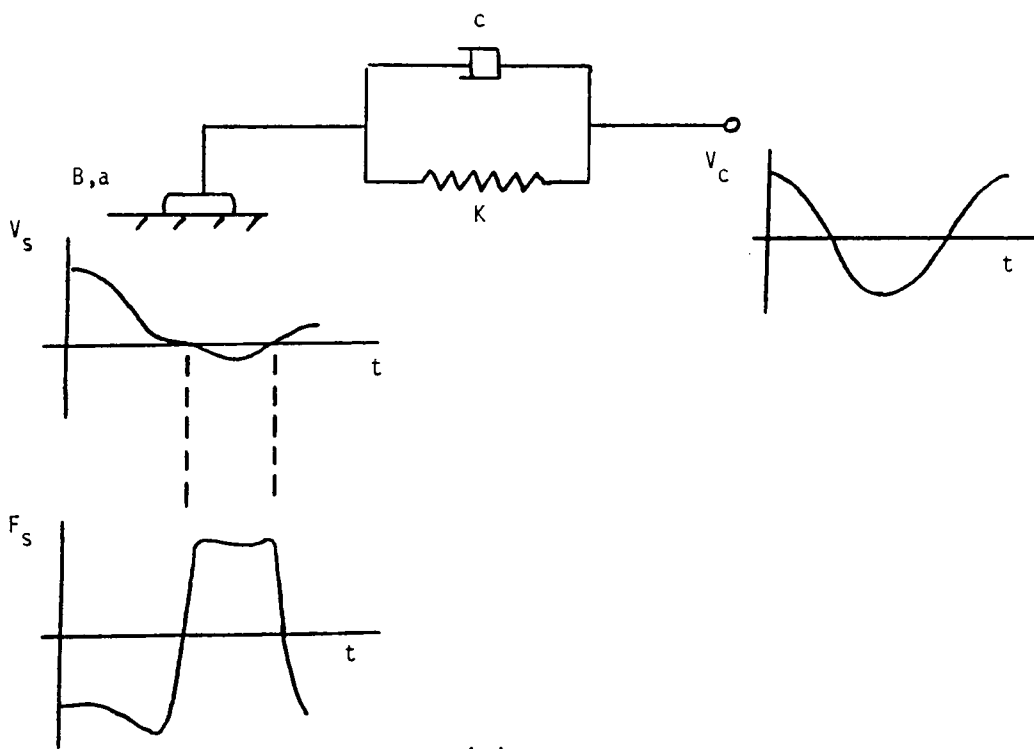
$$\frac{dV_s}{dt} = \frac{K(V_c - V_s) + c\dot{V}_c}{c + \frac{dF_s}{dV_s}}$$

In this equation, dF_s/dV_s is the rate of change of sliding friction with sliding velocity. This can be obtained by differentiating the hysteretic theory given above or measured experimentally. Footprint transit time, t , is taken as the independent variable. The time is equivalent to location in the footprint for steady rolling.

Figure 15(b) shows a schematic diagram of the sliding tread model and its function in converting a frictionless sliding velocity distribution into sliding velocity influenced by friction.



(a)



(b)

Figure 15

IV. FUTURE DIRECTIONS FOR CONTACT SOLUTION ALGORITHMS

As in other areas of solid mechanics, future research on contact problems will be directed towards obtaining solutions valid for large deformations. The finite-element method seems particularly well suited for application to contact problems and special elements have already been developed for this purpose. Continuum mechanics research on contact problems should not be neglected, however. A large-deformation contact solution in terms of continuous functions will prove valuable in the analysis of contact with friction and the assessment of friction laws now being proposed for deformable bodies.

A true contact problem is one in which the contact boundary and interfacial pressure distributions are unknown *a priori*. At present, it appears that such problems will be displacement prescribed: Deflection of the body toward the contact surface is specified and integration of the calculated normal component of the contact pressure gives the resultant load. Some effort should be directed towards a load-specified contact problem, perhaps utilizing the principle of stationary complementary energy to calculate the interfacial pressure distribution subject to the prescribed load constraint. Validated solutions for frictionless contact are essential prior to including the effect of friction on the contact solution.

In the analysis of friction, it seems that the study of rolling contact as a steady-state problem has much to offer. As friction is an interactive phenomenon, at least in regard to sliding velocity, sophisticated algorithms are needed to generate the frictional contact solution from the solution for frictionless contact (which will undoubtedly be the starting point).

The following schematic, figure 16, outlines a progression of research on contact problems. Linear sliding contact is excluded from the outline as this is usually a transient situation leading to accelerated wear and abrasion. Much more will be gained by research focused on rolling contact which will, in any case, include sliding.

PROGRESSION OF RESEARCH ON FRICTIONAL CONTACT PROBLEMS

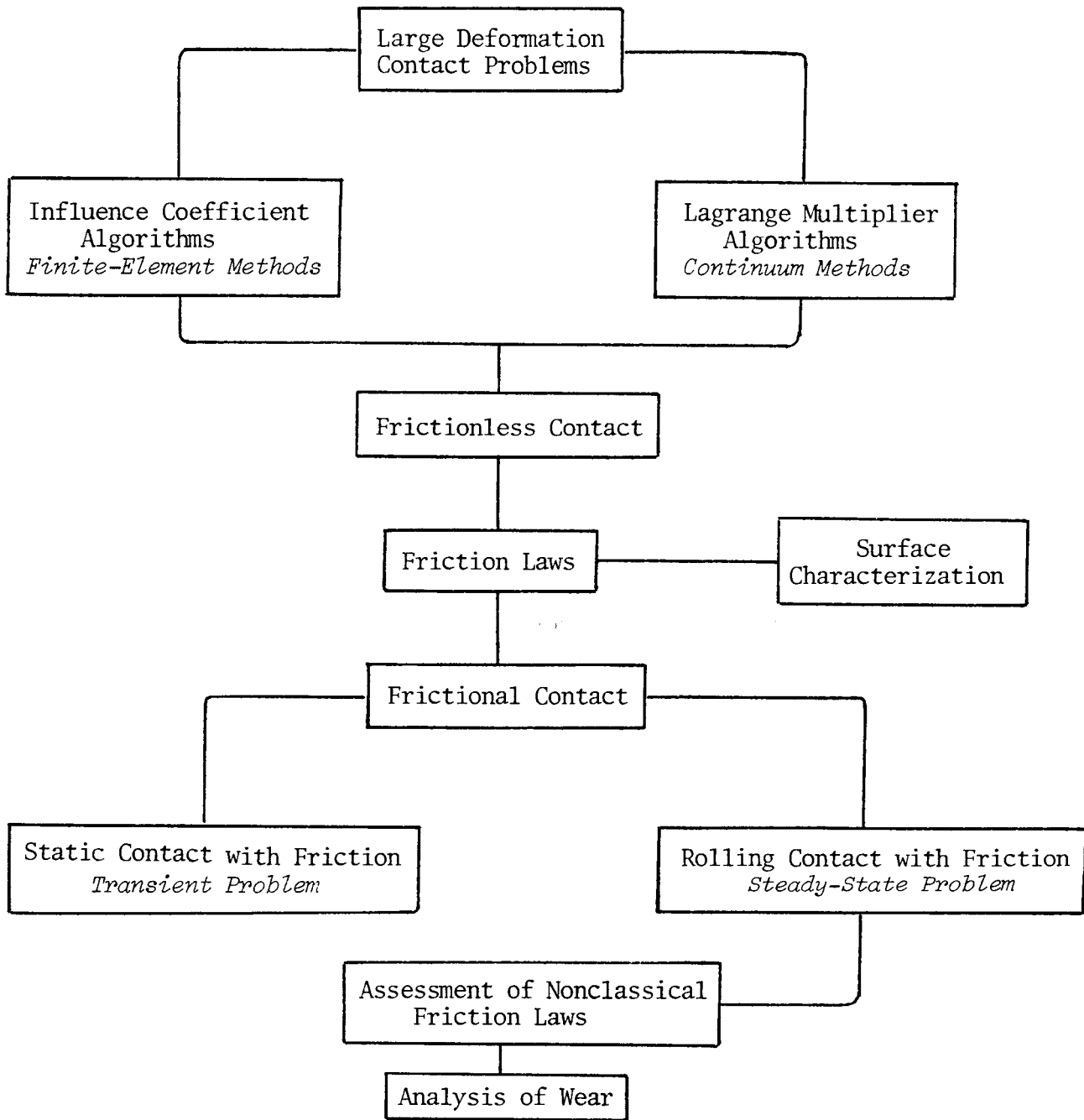


Figure 16

REFERENCES

1. Tillerson, J.R.; and Haisler, W.E.: SAMMSOR II - A Finite Element Program to Determine Stiffness and Mass Matrices of Shells of Revolution. TEES-RPT-70-18, Texas A&M University, Oct. 1970.
2. Haisler, W.E.; and Stricklin, J.A.: SNASOR II - A Finite Element Program for the Static Nonlinear Analysis of Shells of Revolution. TEES-RPT-70-20, Texas A&M University, Oct. 1970.
3. Schapery, R.A.; and Tielking, J.T.: Investigation of Tire-Pavement Interaction During Maneuvering: Theory and Results. Report No. FHWA-RD-78-72, Federal Highway Administration, June 1977.
4. Tielking, J.T.; and Schapery, R.A.: A Method for Shell Contact Analysis. Computer Methods in Applied Mechanics and Engineering, Vol. 26, No. 2, pp. 181-195, May 1981.
5. Green, A.E.; and Adkins, J.E.: Large Elastic Deformations. Oxford University Press, London, 1960.
6. Levinson, M.: The Application of the Principle of Stationary Potential Energy to Some Problems in Finite Elasticity. Journal of Applied Mechanics, Vol. 32, No. 3, 1965, pp. 656-660.
7. Blatz, P.J.; and Ko, W.L.: Application of Finite Elastic Theory to the Deformation of Rubbery Materials. Transactions of the Society of Rheology, Vol. 6, 1962, pp. 223-261.
8. Beveridge, G.S-G.; and Schecter, R.S.: Optimization: Theory and Practice. McGraw-Hill, 1970, p. 264.
9. Treloar, L.R.G.: The Physics of Rubber Elasticity, 2nd edition. Oxford University Press, London, 1958, p. 74.
10. Rothert, H.; Idelberger, H.; Jacobi, W.; and Niemann, L.: On Geometrically Nonlinear Contact Problems with Friction. Computer Methods in Applied Mechanics and Engineering, Vol. 51, 1985, pp. 139-155.
11. Oden, J.T.; and Pires, E.B.: Nonlocal and Nonlinear Friction Laws and Variational Principles for Contact Problems in Elasticity. Journal of Applied Mechanics, Vol. 50, No. 1, March 1983, pp. 67-76.
12. Schapery, R.A.: Analytical Models for the Deformation and Adhesion Components of Rubber Friction. Tire Science and Technology, Vol. 6, No. 1, February 1978, pp. 3-47.

EXPLOITING SYMMETRIES IN THE MODELING
AND ANALYSIS OF TIRES

Ahmed K. Noor, C. M. Andersen, and John A. Tanner
NASA Langley Research Center
Hampton, Virginia 23665

INTRODUCTION

In recent years nonlinear analysis of static and dynamic problems has become the focus of intense research efforts. This endeavor has prompted the development of versatile and powerful finite-element discretization methods as well as of improved numerical methods and software systems for nonlinear static and dynamic analysis of structures and solids. One of the most challenging applications of computational structural mechanics is the numerical simulation of the response of aircraft tires during taxi, takeoff and landing operations. The commonly used models for predicting the tire response are reviewed in Refs. 1 to 3. Figure 1 lists some of the difficulties encountered in the modeling and analysis of tires and their implications.

First, the tire is a composite structure composed of rubber and textile constituents which exhibit anisotropic nonhomogeneous material properties. The laminated carcass of the aircraft tire is thick enough to allow significant transverse shear deformation. *Second*, the tire geometry is complicated and due to the presence of unavoidable imperfections, the cross section is unsymmetric; and *third*, the tire is subjected to inflation pressure and to a variety of unsymmetric mechanical and thermal loads which can result in large structural rotations and deformation, as well as to a variation in the characteristics of the tire constituents. Moreover, the detailed stress and temperature distribution in the tire may require the use of three-dimensional finite elements in certain regions of the tire.

The aforementioned difficulties make the computational expense of the numerical simulation of the tire response prohibitive. Hence, the need for the development of modeling techniques and analysis methods to reduce this expense. Among the modeling strategies which show promise in reducing the cost is the exploitation of symmetries and *quasi-symmetries* exhibited by the tire response.

DIFFICULTY	IMPLICATION
<ul style="list-style-type: none"> o <u>TIRE SHAPE</u> - COMPLEX GEOMETRY - IMPERFECTIONS 	
<ul style="list-style-type: none"> o <u>TIRE MATERIAL AND CONSTRUCTION</u> - ANISOTROPY - NONHOMOGENEITY 	<ul style="list-style-type: none"> o LARGE MODEL SIZE o COMPUTATIONALLY EXPENSIVE
<ul style="list-style-type: none"> o <u>TIRE LOADS</u> - UNSYMMETRY 	

Figure 1

OBJECTIVES AND SCOPE

The objectives of this paper are listed in Figure 2. They are

- 1) To review the different types of symmetry exhibited by the tire response
- 2) To present simple and efficient computational procedures for reducing the size of the analysis model of tires
- 3) To discuss the potential of the proposed techniques and their application to practical, quasi-symmetric tire problems

To sharpen the focus of the study, discussion is limited to two-dimensional shell models of the tire, with elliptic cross-section and linear material response. The analytical formulation is based on a Sanders-Budiansky-type shell theory with transverse shear deformation, anisotropic material behavior, and geometric nonlinearities (moderate rotations) included (Refs. 4 and 5). Displacement finite-element models are used for the discretization. However, the procedure presented herein is expected to be particularly useful for the analysis of three-dimensional tire models.

OBJECTIVES

- REVIEW SYMMETRIES PRESENT IN TIRES
- PRESENT TECHNIQUES FOR MODEL-SIZE REDUCTIONS IN QUASI-SYMMETRIC PROBLEMS
- DISCUSS POTENTIAL APPLICATIONS OF TECHNIQUES

SCOPE

- TWO-DIMENSIONAL SHELL MODELS
- LINEAR MATERIAL RESPONSE
- ELLIPTIC CROSS SECTION
- DISPLACEMENT F. E. MODELS

Figure 2

SYMMETRIES EXHIBITED BY TIRE RESPONSE

The three types of symmetry commonly exhibited by the tire response are shown in Figs. 3 and 4. Also, the difference between the symmetries of orthotropic and anisotropic tires are illustrated.

The first type of symmetry is the *axial symmetry* exhibited by tires whose geometric, material characteristics, and loading are independent of the circumferential coordinate, i.e., axisymmetric. The response of these tires will also be axisymmetric. An example of this situation is that of a tire subjected to uniform inflation pressure. For an orthotropic tire the generalized displacements u , w and ϕ_1 are axisymmetric and v and ϕ_2 are zero. In contradistinction, the five generalized displacements are nonzero for an anisotropic tire.

Reflection (or mirror) symmetry with respect to coordinate planes is exhibited by the response of orthotropic tires when subjected to loadings that exhibit the same type of symmetry. Anisotropic tires, on the other hand, exhibit *rotational (or inversion) symmetry* with respect to the center of symmetry shown in Fig. 4. If these symmetries are exploited in the finite-element analysis, the size of analysis model for an anisotropic tire is twice that of the corresponding orthotropic tire (see, for example, Refs. 6 and 7).

When the external loading exhibits *periodic (or translational) symmetry*, the tire response also exhibits periodic symmetry. In orthotropic tires this is demonstrated by the presence of more than two planes of reflection symmetry. In anisotropic tires periodic symmetry is demonstrated by the presence of more than two centers of rotational (inversion) symmetry. Again, the size of the analysis model for an anisotropic tire with periodic symmetry is twice that of the corresponding orthotropic tire (see Fig. 4).

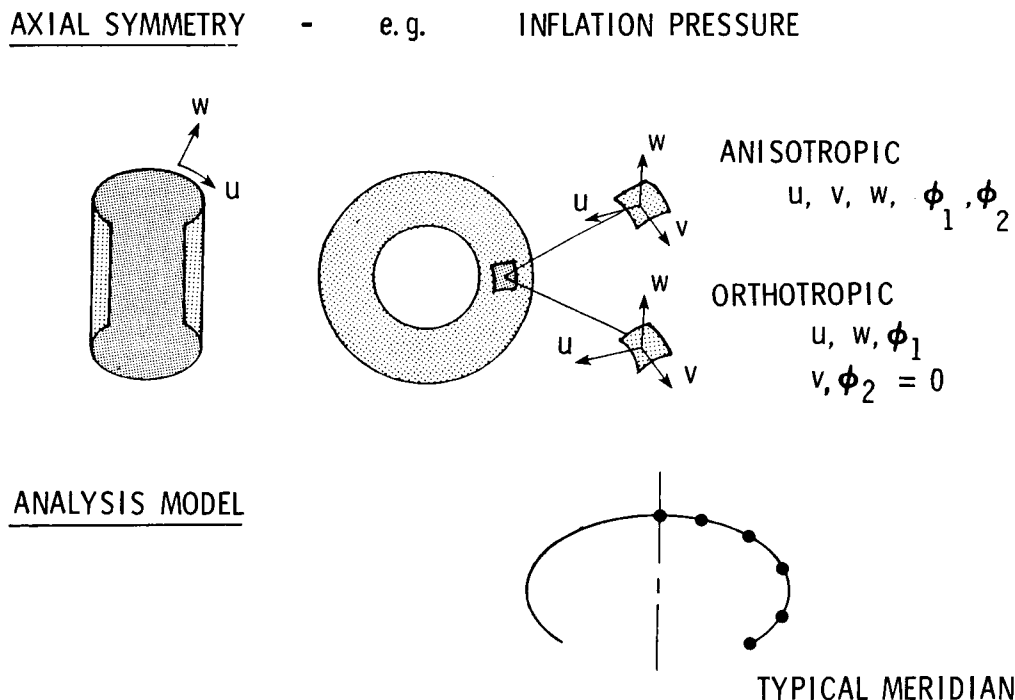
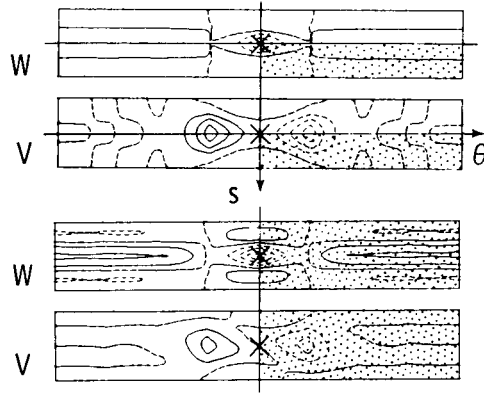
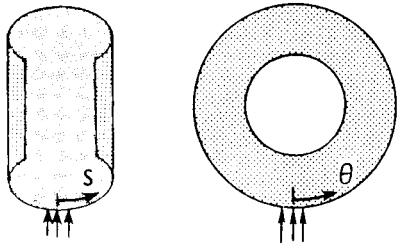


Figure 3

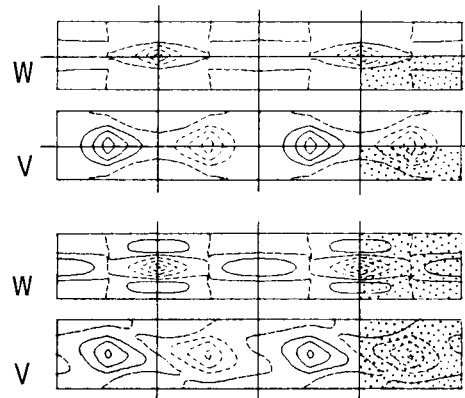
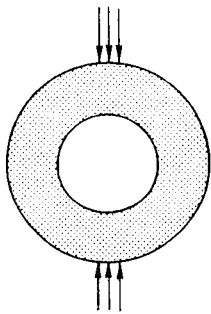
REFLECTION AND
ROTATIONAL SYMMETRIES



ORTHOTROPIC
(REFLECTION AND
ROTATIONAL SYMMETRIES)

ANISOTROPIC
(ROTATIONAL SYMMETRY
ONLY)

PERIODIC SYMMETRY



ORTHOTROPIC

ANISOTROPIC

Note: Shading shows analysis region.

Figure 4

QUASI-SYMMETRIC PROBLEMS

Figure 5 lists a number of quasi-symmetric problems. The three basic quasi-symmetric problems are the ones for which either the tire material, loading, or geometry are not symmetric, but the other two are symmetric. Here material anisotropy is considered to be a source of reflection unsymmetry. The unsymmetry in geometry can be caused by the presence of unsymmetric imperfections in the tire. In a practical situation combination of the three basic types of unsymmetry can exist. In the present study, a computational procedure is presented for reducing the size of the analysis models for quasi-symmetric problems of tires to those of the corresponding symmetric problems.

CASE	MATERIAL	LOADING	GEOMETRY
I	ANISOTROPIC	SYMMETRIC	AXIALLY SYMMETRIC
II	ORTHOTROPIC	UNSYMMETRIC	AXIALLY SYMMETRIC
III	ORTHOTROPIC	SYMMETRIC	UNSYMMETRIC DUE TO IMPERFECTIONS

Figure 5

BASIC IDEA OF MODEL-REDUCTION TECHNIQUE FOR QUASI-SYMMETRIC PROBLEMS

Figure 6 lists the two key elements of the model reduction technique when applied to the finite-element analysis of anisotropic tires with symmetric geometry subjected to symmetric loading. The two elements are: a) decomposition of the stiffness matrix into the sum of an orthotropic and nonorthotropic (anisotropic) parts; and b) successive application of the finite-element method and the classical Rayleigh-Ritz technique. The finite-element method is first used to generate few global approximation vectors (or modes). Then the amplitudes of these modes are computed by using the Rayleigh-Ritz technique.

ANISOTROPIC MATERIALS

- DECOMPOSITION OF MATRICES IN GOVERNING FINITE-ELEMENT EQUATIONS INTO ORTHOTROPIC AND NONORTHOTROPIC PARTS
- SUCCESSIVE APPLICATION OF
 - FINITE-ELEMENT ANALYSIS TO GENERATE A FEW GLOBAL APPROXIMATION VECTORS USING SAME SIZE MODEL AS FOR THE ORTHOTROPIC CASE
 - CLASSICAL RAYLEIGH-RITZ TECHNIQUE TO COMPUTE AMPLITUDES OF APPROXIMATION VECTORS

Figure 6

MATHEMATICAL FORMULATION

Figure 7 outlines the mathematical formulation for the proposed model-reduction technique when applied to linear problems of anisotropic tires. The global stiffness matrix $[K]$ is decomposed into orthotropic and nonorthotropic matrices $[K]_o$ and $[K]_a$, respectively. The nonorthotropic matrix $[K]_a$ is multiplied by a tracing parameter λ which identifies *all* the nonorthotropic material coefficients. The original finite-element equations correspond to the case $\lambda=1$.

The global approximation vectors are selected to be the solution corresponding to $\lambda=0$ (zero nonorthotropic matrix) and its various-order derivatives with respect to λ (path derivatives). The path derivatives are obtained by successive differentiation of the governing finite-element equations with respect to λ . Note that the coefficient matrix appearing on the left-hand sides of the recursion formulas is $[K]_o$, and that the size of the analysis region used in evaluating each of the global approximation vectors is the same as that for the orthotropic case ($\lambda=0$).

The vector $\{X\}_o$ and its path derivatives are now chosen as approximation vectors, and the vector of nodal displacements for the anisotropic tire, $\{X\}$, is expressed as a linear combination of these vectors. A Rayleigh-Ritz technique is used to replace the original finite-element equations by a reduced system of equations in the unknown parameters, $\{\psi\}$, which represent the amplitudes of the global approximation vectors.

GOVERNING FINITE-ELEMENT EQUATIONS

$$\begin{aligned} [K] \{X\} &= \{P\} \\ \text{LET } [K] &= [K]_o + \lambda [K]_a \\ \lambda &= \text{TRACING PARAMETER} \end{aligned}$$

BASIS REDUCTION

$$\{X\} = [\Gamma] \{\psi\}$$

WHERE

$$[\Gamma] = \left[\{X\}_o, \left\{ \frac{\partial X}{\partial \lambda} \right\}_o, \dots \right]$$

$$\{\psi\} = \text{VECTOR OF AMPLITUDES OF MODES}$$

GLOBAL APPROXIMATION VECTORS

$$\left. \begin{aligned} [K]_o \{X\}_o &= \{P\} \\ [K]_o \left\{ \frac{\partial X}{\partial \lambda} \right\}_o &= -[K]_a \{X\}_o \\ &\vdots \end{aligned} \right\} \begin{array}{l} \text{OBTAINED FROM SAME SIZE MODEL} \\ \text{AS FOR THE ORTHOTROPIC CASE} \end{array}$$

REDUCED EQUATIONS (VIA RAYLEIGH-RITZ TECHNIQUE)

$$[\Gamma]^T [K] [\Gamma] \{\psi\} = [\Gamma]^T \{P\}$$

Figure 7

APPLICATION TO ANISOTROPIC TIRES

As a first application of the proposed model-reduction technique consider the anisotropic tire subjected to the symmetric localized loading shown in Figure 8. As shown in Figure 4, the tire response exhibits rotational (inversion) symmetry. It does not exhibit reflection symmetry, and therefore, the analysis model consists of half the tire. As can be seen from the contour plots of Figure 8, all the global approximation vectors exhibit reflection symmetry (and antisymmetry) and therefore, they can be obtained by analyzing only one quadrant of the tire (same size model as that used for analyzing the corresponding orthotropic tire).

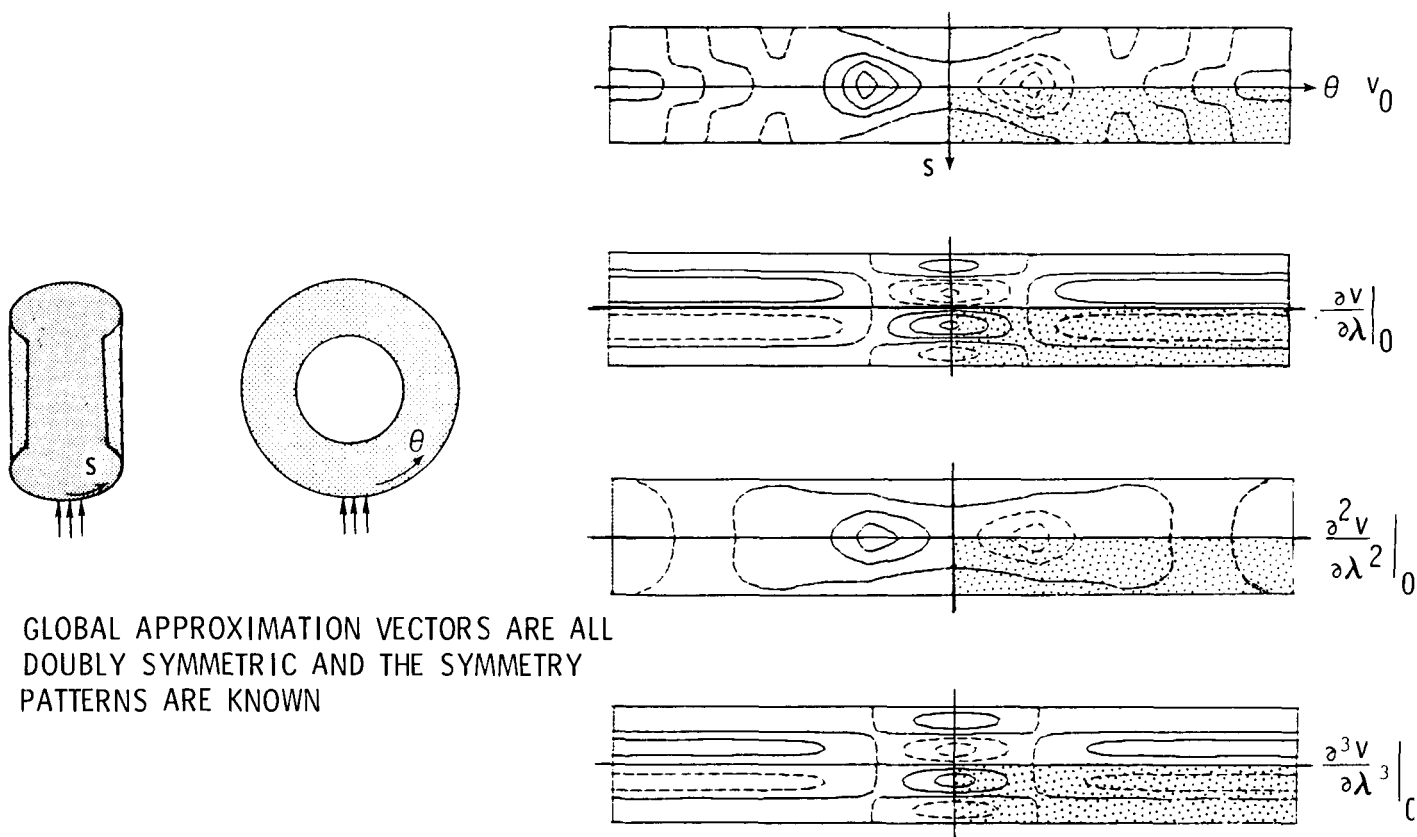
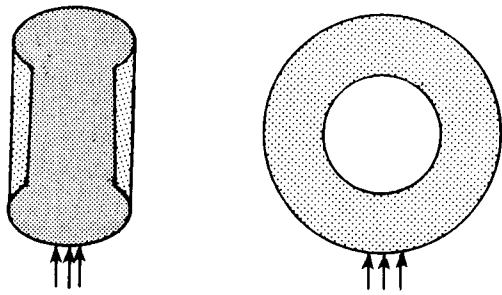


Figure 8

ACCURACY AND CONVERGENCE OF SOLUTIONS OBTAINED
BY PROPOSED TECHNIQUE

An indication of the accuracy and convergence of the solutions obtained by the proposed model reduction technique is given in Fig. 9. The standard of comparison is taken to be the direct finite-element solution of the anisotropic tire. As can be seen from Fig. 9, the solutions obtained by the proposed technique are highly accurate even when a small number of approximation vectors are used. Numerical experiments have shown that for highly anisotropic tires no more than five approximation vectors are needed.



$U = \text{TOTAL STRAIN ENERGY}$

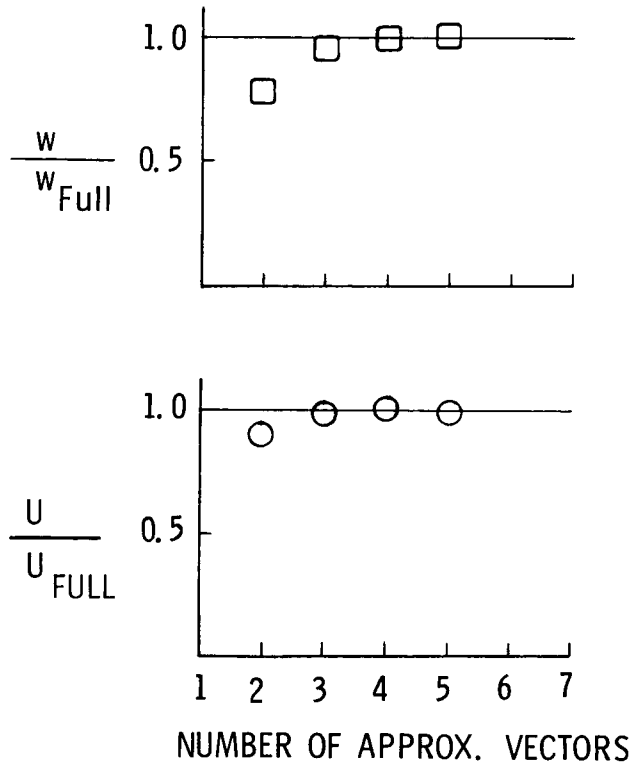


Figure 9

APPLICATION OF MODEL-REDUCTION TECHNIQUE TO PREDICTING NONLINEAR
RESPONSE OF ORTHOTROPIC TIRES SUBJECTED TO ASYMMETRIC LOADING

As another application of the proposed model-reduction technique, consider the orthotropic tire subjected to the localized loading shown in Fig. 10. The analysis model consists of one quadrant of the tire. For linear problems, the decomposition of the loading into symmetric and antisymmetric components and the consequent reduction of the size of the analysis model to one-octant of the tire are well known. However, it is generally assumed that such a decomposition is not useful for non-linear problems in which the principle of superposition is not applicable.

The foregoing model-reduction technique can be used to reduce the size of the analysis model to one octant of the tire. This is accomplished by: 1) decomposition of the given loading into symmetric and antisymmetric components with load parameters p_1 and p_2 , respectively; and 2) use of the multiple-parameter reduction technique described in Refs. 8 and 9. The vector of nodal displacements of the tire is approximated by a linear combination of few global vectors or modes. These vectors are selected to be the various-order derivatives of the displacement vector with respect to load parameters p_1 and p_2 . Each of these vectors is evaluated at $p_1=p_2=0$. Figure 10 shows that the global approximation vectors exhibit periodic symmetries (and/or antisymmetries), and therefore, each vector can be evaluated using only one octant of the tire.

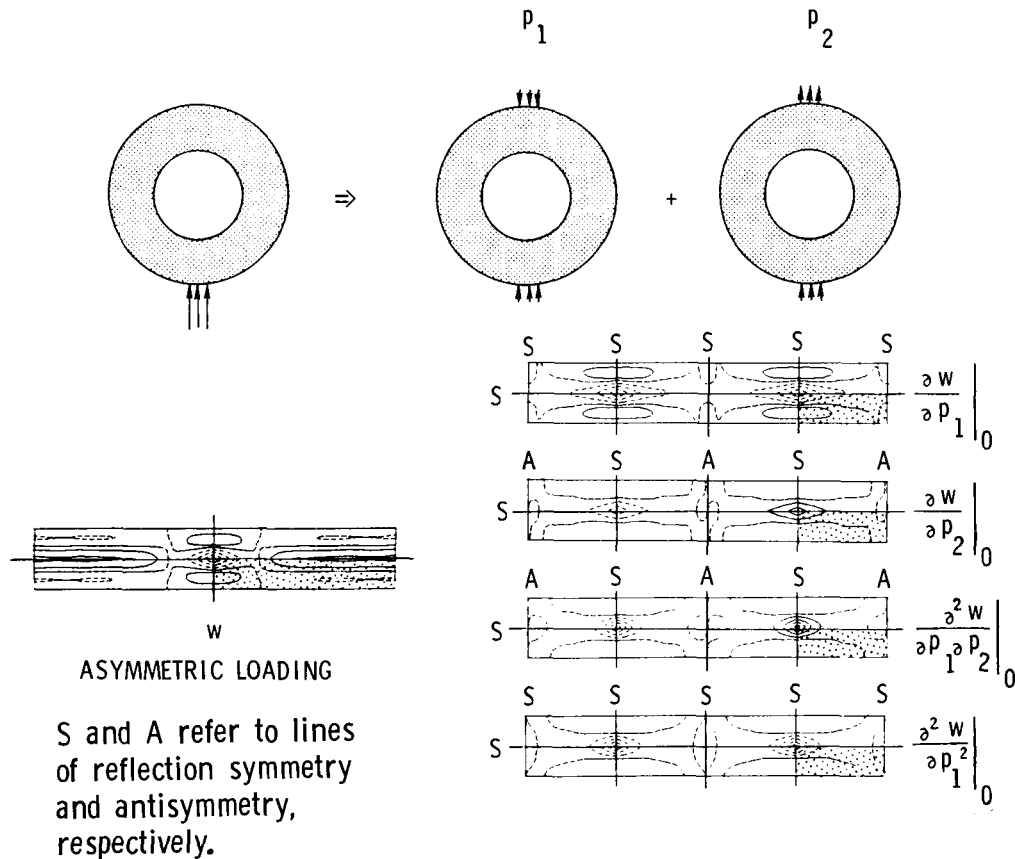


Figure 10

POTENTIAL OF PROPOSED MODEL-REDUCTION TECHNIQUE

The proposed model reduction technique appears to have high potential for analysis of practical tire problems. In particular, in the presence of combinations of unsymmetries in the material (viz. anisotropy), the geometry or loading on the tire, the same size model can be used as for symmetric material (viz., orthotropic), geometry and loading (Fig. 11). This is accomplished by the introduction of a tracing parameter for each of these unsymmetric effects; and the successive application of a reduction method with each of these parameters. The global approximation vectors are selected to be the various-order derivatives with respect to the tracing parameters.

- ASYMMETRIC LOADING AND ANISOTROPY
- ANISOTROPY AND INITIAL UNSYMMETRIC IMPERFECTIONS
- ASYMMETRIC LOADING AND INITIAL UNSYMMETRIC IMPERFECTIONS
- THREE-DIMENSIONAL MODELS
- MIXED FINITE-ELEMENT MODELS

Figure 11

SUMMARY

In summary, a computational procedure is presented for reducing the size of the analysis models of tires having unsymmetric material, geometry and/or loading. The two key elements of the procedure when applied to anisotropic tires are: a) decomposition of the stiffness matrix into the sum of an orthotropic and nonorthotropic parts; and b) successive application of the finite-element method and the classical Rayleigh-Ritz technique. The finite-element method is first used to generate *few* global approximation vectors (or modes). Then the amplitudes of these modes are computed by using the Rayleigh-Ritz technique.

The proposed technique has high potential for handling practical tire problems with anisotropic materials, unsymmetric imperfections and asymmetric loading. It is also particularly useful for use with three-dimensional finite-element models of tires.

- MODEL-SIZE REDUCTION PROCEDURE PRESENTED FOR ANALYSIS OF TIRES BASED ON
 - DECOMPOSITION OF MATRICES
 - SUCCESSIVE APPLICATIONS OF FINITE ELEMENTS AND RAYLEIGH-RITZ TECHNIQUES
- PROPOSED TECHNIQUE HAS HIGH POTENTIAL FOR HANDLING
 - ASYMMETRIC LOADING
 - ANISOTROPIC MATERIALS
 - UNSYMMETRIC IMPERFECTIONS
 - THREE DIMENSIONAL MODELS

Figure 12

REFERENCES

1. Clark, S. K. (ed.), "Mechanics of Pneumatic Tires," U.S. Dept. of Transportation, National Highway Traffic Safety Administration, Washington, D.C., 1981.
2. Tanner, J. A. (compiler), "Tire Modeling," NASA CP-2264, 1983.
3. Noor, A. K. and Tanner, J. A., "Advances and Trends in the Development of Computational Models for Tires," Computers and Structures, 1985.
4. Sanders, J. L., "Nonlinear Theories for Thin Shells," Quarterly of Applied Mathematics, Vol. 21, April 1963, pp. 21-36.
5. Budiansky, B., "Notes on Nonlinear Shell Theory," Journal of Applied Mechanics, Vol. 35, June 1968, pp. 393-401.
6. Noor, A. K. and Camin, R. A., "Symmetry Considerations for Anisotropic Shells," Computer Methods in Applied Mechanics and Engineering, Vol. 9, 1976, pp. 317-335.
7. Noor, A. K., Mathers, M. D. and Anderson, M. S., "Exploiting Symmetries for Efficient Postbuckling Analysis of Composite Plates," AIAA Journal, Vol. 15, January 1977, pp. 24-32.
8. Noor, A. K. and Peters, J. M., "Recent Advances in Reduction Methods for Instability Analysis of Structures," Computers and Structures, Vol. 16, 1983, pp. 67-80.
9. Noor, A. K., Andersen, C. M. and Tanner, J. A., "Mixed Models and Reduction Techniques for Large Rotation, Nonlinear Analysis of Shells of Revolution with Application to Tires," NASA TP-2343, October 1984.

PANEL DISCUSSION, Wednesday, June 19 - TIRE MODELING

J. A. Tanner, NASA Langley Research Center: You've heard five papers this morning in the general area of tire modeling. However, I believe that the content of the papers is actually broader than tire modeling and that we are really dealing with the more general issues associated with computational structural mechanics. I was rushing some of the authors this morning, trying to rush them through their presentations, and I am concerned that they may not have made all the points they wanted to make. In an effort to make amends, I would like to ask each author to take about 2 or 3 minutes to summarize any points that he would like to emphasize and to address points not made in his presentation. After the summary statements, the floor will be opened for additional questions. I am going to start immediately to my right with Dr. Tabaddor who gave the first paper this morning.

Farhad Tabaddor, B. F. Goodrich Company: I would just like to reiterate the point on the material properties of composites. There is a great need for efforts in characterization of composites, and I think that theories developed over the past decade or so for rigid composites do not quite apply to soft composites. I would recommend that maybe some sort of grant effort might be initiated by NASA to do this basic work.

Tanner: Dr. Padovan, in your presentation you dealt with the architectural characteristics of the new computers that will be coming on line. I thought maybe you would have some more things you wanted to say on that subject.

Joe Padovan, University of Akron: I think we would like to see the availability of some real number crunching power so that we could really push models to the limit. Secondly, I think that we are all too interested in the local detail information in the model and I think we are not really trying to capture the true essence of what a tire does, and that is to provide good ride characteristics. I think more effort should be put towards characterizing the overall features of the true structure and leave most of the stress analysis to the tire industry where it belongs. Now I am speaking for myself. I think the dynamic characteristics of the tire are most important. More effort has to be put into that kind of modeling. Thirdly, more money has to be put into the effort in general. If you really want major changes in the technologies that have been in place for 40 years then you have to spend a great deal of money.

Tanner: Dr. Oden, in your presentation this morning you spent some time talking about error analysis. The tire industry will not use any analysis tools unless their solution accuracies are verified. I imagine that error analysis is critical to the verification of any modeling strategy. Would you care to comment further on this subject?

J. Tinsley Oden, The Computational Mechanics Company, Inc.: It is gratifying to see the word "error" used more and more in meetings such as this because I believe the users of finite element methods have finally reached a point where they are beginning to demand some assurance as to the reliability and quality of solutions. We have spent 10 years in developing elements, then algorithms, and now machines to handle the algorithms, but how good are the answers? Methods are evolving that will someday allow us to answer that question with some degree of confidence. However, there are methods at our disposal now that are based on estimations of errors that will allow us to incorporate adaptive strategies into very general finite element structural calculations. I heartily support the activity in this area. I think we are going to see some error estimation and adaptivity become an integral part of finite element calculations in the near future.

Tanner: Dr. Tielking, the central theme of your paper was contact algorithms and you presented results from static contact, both with and without friction. I was particularly interested in your last comment, however, when you indicated that future studies ought to be concerned with rolling contact. I thought you might want to expand a little on that thought.

J. T. Tielking, Texas A&M University: Yes, I would like to say a little bit more about the study of contact with friction. Of course, all contact involves friction. The assumption of frictionless contact is analogous to the assumption of a rigid body--we really don't have such structures. I feel that the finite element models certainly do have a place in the study of friction, but friction forces vary continuously through the interface, they depend, of course, on the variations of several variables, typically interfacial pressure, normal pressure, and sliding velocity. It is now recognized that there is sliding velocity everywhere in the tire contact region. The assumption of regions of adhesion and sliding are good for approximations, but not for more accurate results. I feel that the continuous sliding models should not be neglected. One of my motivations for doing that plain strain cylinder problem was to get a large

deformation contact solution which could be used in the study of friction. Therefore, I am very pleased to see that you are not restricting the CSM activity to finite element methods and that you will entertain alternative methods of solution. Finally, I appreciate your concern for university funding. Foreign governments have been sending their best students to our universities for a long time and we are very happy to have them, but we would like to be able to fund our own students. I think organizations such as NASA could certainly do a great deal to see that students from our own country can be funded as well as the top students from foreign countries.

Tanner: Now I will let the first two authors on the last paper have their say.

A. K. Noor, George Washington University: I would like to make a few comments about the paper. I think first of all this was really a study that aims at developing a general computational strategy. In very general terms, when we are confronted with a complex problem, we generate the solution to that problem starting from a simpler problem or perhaps even from a hierarchy of simpler problems. I think this is very much an engineering-type approach. These simpler problems will each be associated with a control parameter. What we are doing is essentially using a perturbation technique. What we have heard in the last presentation was an extension of the classical perturbation technique which is limited to small values of the perturbation parameter. The technique that we have developed is not limited to small values of the perturbation parameter; it works well even if you have very large values for the perturbation parameter. Our example was for a highly anisotropic tire. The perturbation parameter was indicative of the degree of anisotropy. The technique, I think, can be essentially summed up as a combination of operator splitting, where we did the splitting on the matrices, and a multiple parameter extended perturbation. The other comment which I would like to make is that you get the sensitivity information free because all that information is needed in the daily use of the equations. In other words, the sensitivity information is a by product of the solution algorithm. The final comment is that the CPU time was reduced by a factor of eight or more for the problems that you have seen but that reduction factor can be much higher for dynamic problems.

C. M. Anderson, College of William and Mary: Well, to paraphrase our paper just one other way, if the geometry, material properties, and loading in a structure exhibit a kind of symmetry, then you may well expect the solution will exhibit that same symmetry. It is very easy to take that symmetry into account and reduce your model size. What we are saying is that one can now introduce a higher symmetry group into the problem. For nonlinear problems which do not have that full symmetry, one can find basis functions which represent the higher symmetry group. The solution that we are looking for will be a linear combination of those basis functions. Therefore, we can still achieve the reduced model size even though the full symmetry is not present in the final solution that we are looking for.

Tanner: I have a few closing comments. For the past 2 years, NASA and the U.S. tire industry have been involved in a joint program called the National Tire Modeling Program. The program provides a forum for technical discussions without infringing on the proprietary rights of the individual tire companies. The objective of the program is to develop tire analysis tools to help streamline the tire design process. The number of tire industry representatives involved in this workshop is an indication of the strong industry support for the program.

The last point that I want to make involves the relationship between tire modeling studies and computational structural mechanics in general. The paper that Dr. Tabaddor presented this morning gives an indication of the material property concerns that exist in the tire industry. Although some of the concerns expressed by Dr. Tabaddor are unique to the rubber-cord composites, I believe that many of his material property concerns are common to both soft and rigid composites. I think that many of the technical challenges that we face in our tire modeling efforts are common to each of the disciplines represented here today. This CSM workshop is a great way of getting a lot of people together to find out what we have in common in our technical pursuits. We should do this more often.

Any questions from the audience?

Barna A. Szabo, Washington University: I'd like to address my question to Prof. Oden. You mentioned the error estimation which uses the error indicators that you presented. I understand that you are computing the contribution of error in the energy norm of each element to the total error of approximation. You would

like to see that each element contributes about the same error if the mesh is properly designed. We found that in some cases we can have very small errors in the energy norm, something of less than 1 percent and, yet, have more than 20 percent error in equilibrium. For example, if you cut the structure off the supports and recompute the reactions from the finite element solution using the direct method, we still can have a fairly large error in equilibrium. It is important to look at the error in energy norm but it is also important to check other quantities of interest. Have you observed the same thing? I am particularly fond of the equilibrium check because it can be applied to nonlinear problems as well as to linear problems and also it is meaningful to engineers. One last comment, if an element or group of elements is found to be out of equilibrium then the possibility of pollution of that error arises, whereas if the group of elements is in equilibrium then according to St. Venant's principle you would expect the error to be localized. This is a qualitative view, but, nevertheless, it has a certain convincing flavor to it that engineers will appreciate.

Oden: The presence or absence of equilibrium globally has to do with the residual in a solution when you are talking about linear or nonlinear elliptic problems. The calculation of the error indicator does compute a local residual. It computes a local residual that enters the right-hand side of the equation. The problem is that the residual by itself is not an indication of error. If your residual is large, chances are the error is large, but the converse is not true. You can have a system of forces in equilibrium and have an enormous error. That is the complicating feature of these kinds of error estimation. One must select an indication of the error that has the right asymptotic behavior such that as the indicator goes to zero, so also does the error go to zero. The energy norm exhibits this characteristic. The points you raise are subtle but very important issues in calculating error estimators. These points are the complicating feature in this particular kind of strategy because one has to design an error indicator with the proper asymptotic properties. There are many methods for estimating local error, but the one I am speaking of now, while expensive, is the best in that it overcomes the very problems that you are referring to. It produces a robust and accurate estimation of the local error which does, in fact, vanish at a rate equal to the vanishing of the residual as the mesh is refined.

Gerald Goudreau, Lawrence Livermore National Laboratory: I'm new to tire modeling and the one thing I didn't get out of this morning was a feeling for how you get your composite material models of a tire. I get a feeling that you go out and measure rubber properties, even get finite deformation or finite strain models of rubber, you do load deflection tests on cords and, yet, somehow this is all put together into some kind of a model which may be orthotropic, maybe anisotropic, but who's doing what? Anyone can respond.

Joe Padovan: Measurements of the material properties of tire constituents have been going on for many years. The industry does not measure rubber properties and cord properties separately but rather the aggregate cord/rubber composite. Typically, the rule that has been used for mixtures is Halpin-Tsai, although this does not fully apply in a nonlinear setting like the tire.

Goudreau: Are these small strain models then?

Padovan: There are not really any large deformation constitutive models except for some very isolated materials. In the case of rubber/cord composites, there certainly are none yet. Basically what you do is assume Halpin-Tsai or whatever linear model you are using works and then assume large strains and repeat Kirchhoff's assumptions on the other side of the equation.

Goudreau: How are you characterizing this anisotropy?

Padovan: The anisotropy is handled through standard laminate theory.

Goudreau: You're testing things lamina by lamina then?

Padovan: Yes, lamina by lamina.

Marion G. Pottinger, B. F. Goodrich Company: I would like to make some comments. Structural mechanics love to calculate the tire stresses and strains, but tire users are concerned with performance characteristics which are very different things. Since the cords are pretty big and you are treating the tire as a continuum, there are some definitive limitations on what you can do. We are worrying about being exact, but how does this exactness fit in with the tried and proven design rules of thumb. The old design rules of thumb are very

important to design people and so are the methods of qualitative characterization. Finally, I would like to mention the question of combining the analytical and the experimental and the question of experimental verification. Not only do you have to satisfy yourself that things really work, you also have to satisfy lawyers.

Richard B. Nelson, U.C.L.A.: I am a user of finite elements, but I also use cars and tires. My experience is that heat, abuse, and, delamination are what kill tires. I wonder if there has been attention given in the industry to the need to develop models which are capable of predicting heat build-up and what that does in the way of generating premature tire delaminations.

Tanner: Of course, we are all extremely interested in the thermal characteristics of the tire constituents. As Dr. Tabaddor pointed out this morning, the constituents are highly sensitive to temperature. In the area of aircraft tires, we are very concerned about the transient temperatures that are generated during normal taxi operations. Truck and automobile tire people have similar concerns except that they deal with rolling for many many hundreds of miles at a time at stabilized temperatures.

Dr. Tabaddor: With respect to Dr. Nelson's question, experimentally, of course, there is much concern about the effect of the temperature on material properties, especially fatigue and fracture properties. There is a significant degradation of these qualities with increasing temperature. The analytical model will also be extremely complex because you have a mechanical model and rolling contact while viscoelasticity generates heat and that heat in turn affects the tire material properties.

Dr. Tielking: One objective of the rolling contact analysis which includes friction and viscoelasticity is to get the energy dissipation due to tire flexing into a heat conduction analysis. There is some preliminary work already under way in the National Tire Modeling Program that's involved in transient heat conduction analysis for rolling tires. Sam Clark has been involved in that to a great extent both experimentally and computationally over the last couple of years.

Joop Nagtegaal, MARC Analysis Research Corp.: I have had quite a bit of experience with the tire industry. I would like to make a comment on the remarks from

the gentleman of Goodrich. In the late 70's when I was dealing with several tire industries on another continent, they would pose a simple problem that could be solved, but then say that that is not the whole problem. I would have to account first for one effect and then another until, finally, the problem would become unsolvable. Now tire problems are extremely complicated, I understand that, but I think it would be extremely useful if this National Tire Modeling Program could come up with a clear set of problems to solve. Maybe this is already being done because I see a lot of positive contributions here which are very different from many years ago.



Report Documentation Page

1. Report No. NASA CP- 3034, Part 1		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle Computational Methods for Structural Mechanics and Dynamics				5. Report Date May 1989	
				6. Performing Organization Code	
7. Author(s) W. Jefferson Stroud, Jerrold M. Housner, John A. Tanner, and Robert J. Hayduk, Editors				8. Performing Organization Report No. L-16560	
				10. Work Unit No. 505-63-01-10	
9. Performing Organization Name and Address NASA Langley Research Center Hampton, VA 23665-5225				11. Contract or Grant No.	
				13. Type of Report and Period Covered Conference Publication	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001				14. Sponsoring Agency Code	
				15. Supplementary Notes	
16. Abstract The conference publication contains the proceedings of the Workshop on Computational Methods for Structural Mechanics and Dynamics held at NASA Langley Research Center, June 19-21, 1985. The Workshop was organized into the following four sessions: (1) Local/Global Nonlinear Stress Analysis (2) Tire Modeling (3) Transient Dynamics (4) Multibody Dynamics Transcriptions of discussions are also included.					
17. Key Words (Suggested by Author(s)) Local/Global Parallel Processing Global/Local Tire Modeling Transient Dynamics Multibody Dynamics			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 342	22. Price A15