# 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 shear 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 $\bar{\sigma} \neq \sigma_o$. 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.

C-2

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 <u>definition</u> 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?

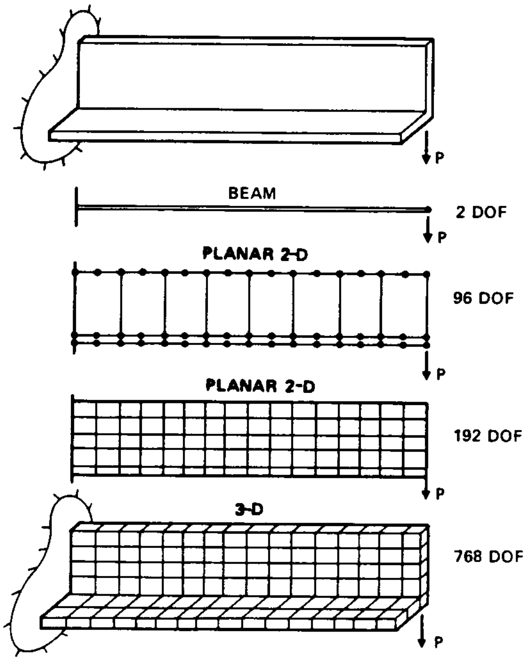
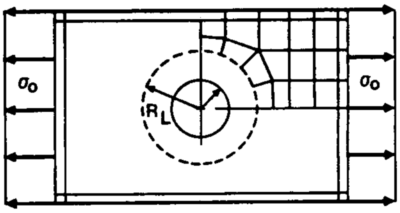


Figure 2

# NEAR-FIELD MODELING

Singularities/Stress Concentrations

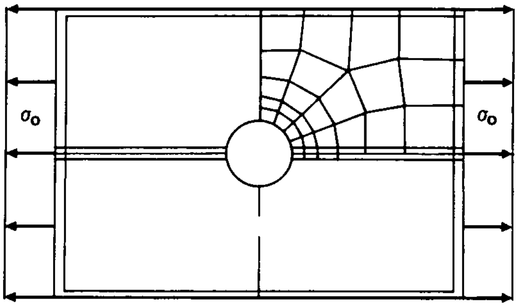
- Finite-Element Model with Near-Field Analytical Solution



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

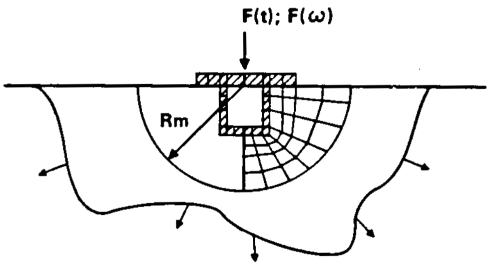
Figure 4

# FAR-FIELD MODELING

**Finite Structure Imbedded in Infinite Continuum**

**– Finite Element Model with Far Field**

**Analytical Solution**



**Analytical Solution(s) Required**

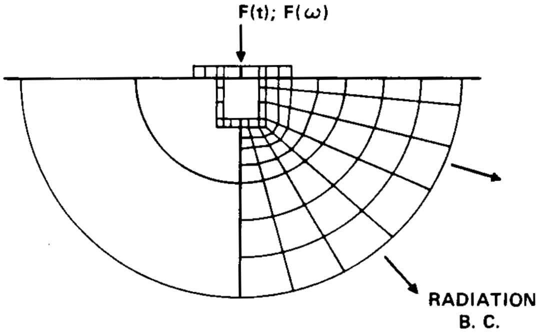
**Capable of Being Evaluated and Utilized**

**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

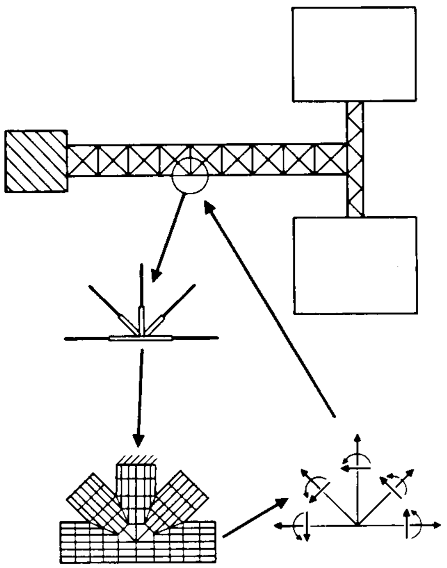


Figure 7

- Surrounding Substructure

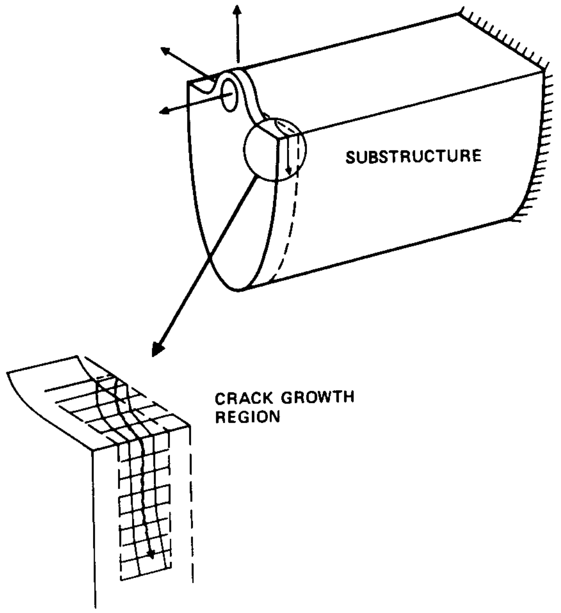


Figure 8

# ADAPTIVE FINITE ELEMENT MODELING

### BLADE-STIFFENED PANEL WITH DISCONTINUOUS STIFFENER

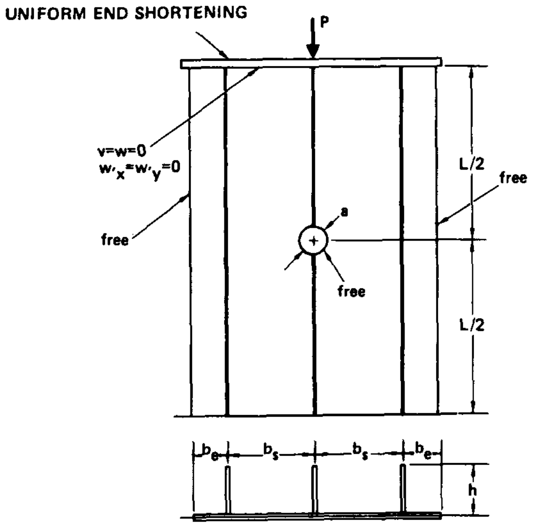


Figure 9

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

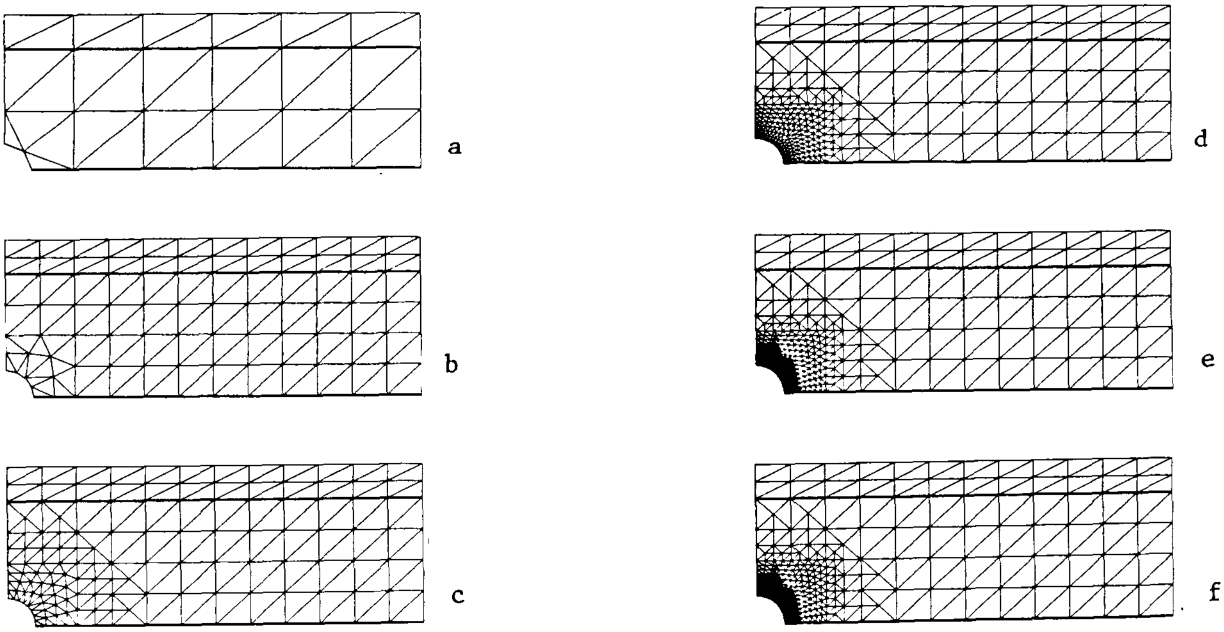


Figure 10