N87-22785

THE NESSUS FINITE-ELEMENT CODE*

J.B. Dias, J.C. Nagtegaal, and S. Nakazawa
MARC Analysis Research Corporation
Palo Alto, California

The NESSUS finite element code is being developed by MARC Analysis Research Corporation as part of the probabilistic structural analysis (PSAM) effort, coordinated by Southwest Research Institute for the NASA Lewis Research Center. The objective of this development is to provide a new analysis tool which integrates the structural modeling versatility of a modern finite element code with the latest advances in the area of probabilistic modeling and structural reliability. Version 2.0 of the NESSUS finite element code was released to the members of the PSAM team last February, and is currently being exercised on a set of problems which are representative of typical SSME applications. NESSUS 2.0 allows linear elastostatic and eigenvalue analysis of structures with uncertain geometry, material properties and boundary conditions, which are subjected to a random mechanical and thermal loading environment.

The NESSUS finite element code is a key component in a broader software system consisting of five major modules. NESSUS/EXPERT is an expert system under development at Southwest Research Institute, with the objective of centralizing all component-specific knowledge useful for conducting probabilistic analysis of typical SSME components. NESSUS/FEM contains the finite element code used for the structural analysis and parameter sensitivity evaluation of these components. The task of parametrizing a finite element mesh in terms of the random variables present is facilitated with the use of the probabilistic data preprocessor in NESSUS/PRE. An external database file is used for managing the bulk of the data generated by NESSUS/FEM. To complete the analysis, Southwest Research Institute has developed the probabilistic analysis module NESSUS/FPI, which extracts from the database the information needed for generating probability distributions for the desired response variables.

Probabilistic finite element analysis involves the computation of the effects of small variations of the random variables on the overall response of the structure. Since probabilistic models of realistic structures often require large finite element models parameterized by many random variables, this step is very computation-intensive and will tend to govern the solution cost. The strategy adopted in the NESSUS finite element code is based on an iterative algorithm akin to the modified Newton method. With this approach, all computations on the perturbed system can be performed at the element level, resulting in substantial savings on memory requirements. This approach also eliminates the need to compute and store explicit partial derivatives of the element matrices and vectors, or to assemble these into additional global arrays. As a result, the perturbation of the element parameters can be made independent of element formulation, allowing the introduction of the newest element technology into the existing algorithm with a minimal amount of new software development. Significant improvements in convergence speed have been

---

*Work performed under NASA contract NAS3-24389.

demonstrated with the use of modern algorithms for nonlinear programming, which have been implemented in NESSUS within the past year.

The iterative algorithms implemented in the NESSUS finite element code can be shown to be inherently consistent, but must satisfy known stability conditions in order to achieve convergence. In most cases this will not present a problem, since loss of stability will only occur for very large stiffness changes, which are well beyond the range of uncertainty observed in the physical structure. However, there are ill-conditioned problems for which even seemingly small perturbations will engender large changes in some stiffness terms. This is often the case when constraint equations are present in the problem formulation. Such problems are typically encountered in the analysis of strongly anisotropic or incompressible materials (e.g. composites, rubber mounts and seals), deviatoric rate-independent plasticity, or the thin limit of the Reissner-Mindlin shell theory. A solution to this problem will involve the development of smart adaptive algorithms for selecting the perturbation size in order to ensure convergence of the algorithm.

Significant progress was made over the last year on issues related to the modeling of probabilistic structures and the development of a good user interface. It was soon realized that classical shell theories lack the generality needed to capture localized effects which are often important in the analysis of SSME-related components. This resulted in the development of a new family of finite elements based on an assumed strain formulation and designed to achieve superior performance in bending problems. These elements allow the definition of continuous pressure fields at the surface nodes using a new boundary identification algorithm. Current development efforts are aimed at enhancing the element's thermal strain response and extending the new formulation to problems involving material nonlinearity.

The external database file is used for retaining a permanent record of the parameter sensitivity data generated by the NESSUS code. This database resides in a binary (unformatted) direct-access file, and is structured as a two-way ordered linked list. This type of data structure allows the insertion, deletion and replacement of individual entries without the need to move large blocks of data. It is therefore possible to maintain and expand an existing database with additional results obtained in the course of multiple runs of the NESSUS finite element code. Each database file is self-contained, allowing direct access to the perturbation data from within other modules in the NESSUS system.

Planned developments for the coming year include extending the capabilities of the NESSUS code to perform probabilistic analysis in situations involving material nonlinearity and transient dynamics. This will require tracking separate time-histories for all perturbed problems and transferring perturbation data between increments. The ability to conduct random vibration analysis on uncertain structures is also planned for the near future. The development of probabilistic finite deformation analysis in NESSUS is currently planned for the fourth year of the PSAM effort.

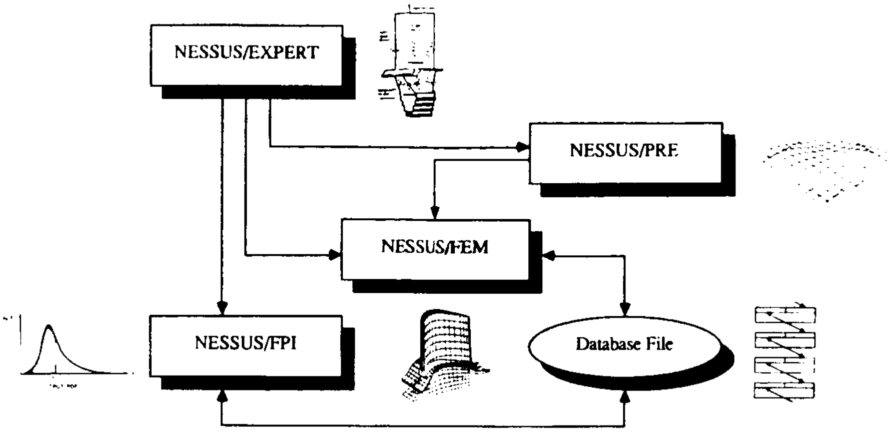

FIGURE 1: AN OVERVIEW OF THE NESSUS CODE

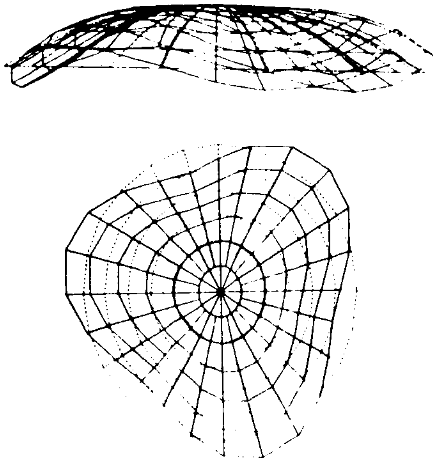

FIGURE 2: UNCOUPLED RANDOM VARIABLES USED TO CHARACTERIZE GEOMETRICAL UNCERTAINTY MAY BE IDENTIFIED WITH SIMPLE GEOMETRY IMPERFECTIONS

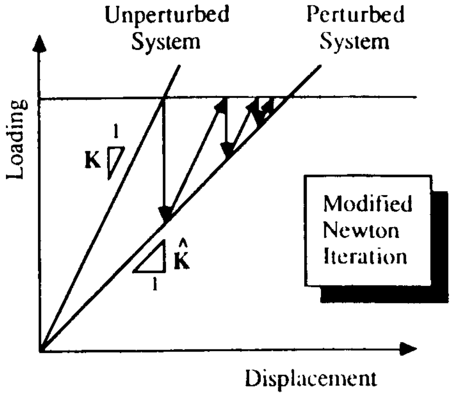

FIGURE 3: GRAPHICAL INTERPRETATION OF MODIFIED NEWTON ITERATION

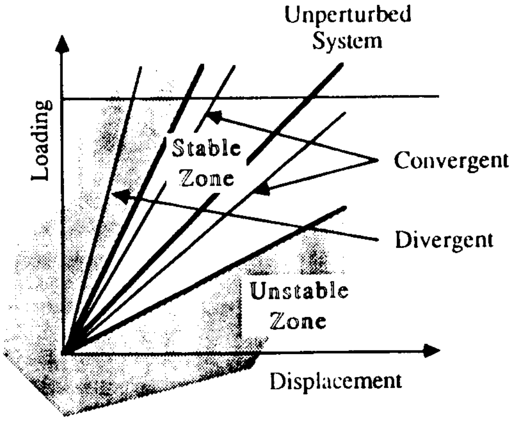

FIGURE 4: STABILITY BOUNDS DEFINE THE ALLOWABLE RANGE OF PERTURBATION SIZES WHICH ARE CONVERGENT WITH THE MODIFIED NEWTON ALGORITHM

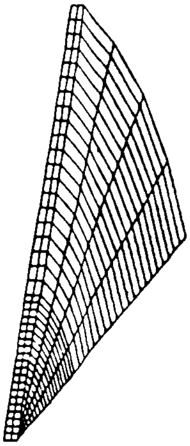

FIGURE 5: LOCALIZED THERMAL AND MECHANICAL EFFECTS CRUCIAL TO THE ANALYSIS OF COMPONENTS SUCH AS THIS BLISTER SPECIMEN MODEL CANNOT BE ADEQUATELY CAPTURED USING SIMPLIFIED SHELL-BASED MODELS

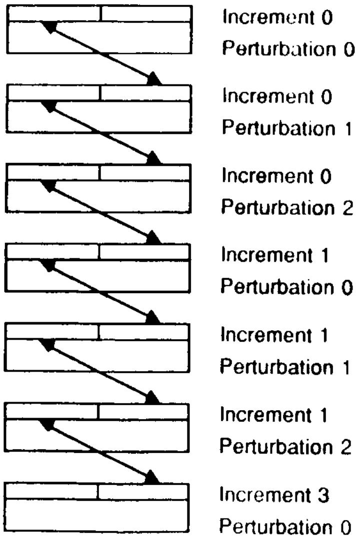

FIGURE 6: AN EXTERNAL DATABASE FILE IS USED TO RETAIN A PERMANENT RECORD OF THE PARAMETER SENSITIVITY DATA GENERATED BY THE NESSUS CODE