NONLINEAR STRUCTURAL ANALYSIS OF COMPLEX ELECTRONIC AND ELECTROMECHANICAL ASSEMBLIES

Johnny H. Biffle and Steven N. Burchett

***Johnny H. Biffle** and **Steven N. Burchett** are in the Engineering Analysis Department of Sandia National Laboratories at Albuquerque, New Mexico. Mr. Biffle is supervisor of Applied Mechanics Division III, where he is responsible for research and development on structural analysis techniques. He joined AT&T in 1966. He received a B.S. in mechanical engineering from Texas Technological University, an M.S. in mechanical engineering from the University of Washington at Seattle, and a Ph.D. in engineering mechanics from the University of Texas at Austin. Mr. Burchett is a member of the technical staff in Applied Mechanics Division I. He joined AT&T in 1968. He received a B.S. and M.S. in mechanical engineering from Oklahoma State University.*

The finite-element technique, which is particularly well suited to problems with complex geometries, has been extensively used in structural mechanics. For the highly nonlinear problem, however, the technique has been limited to problems with a modest number of elements for want of a solution scheme that efficiently uses computer resources. We have found that the conjugate gradient method, taken from the field of optimization, provides the required solution technique. The technique has been adapted in a finite-element formulation for structural mechanics and configured to take advantage of vector processing computer hardware, resulting in the development of two computer programs, the two-dimensional JAC[1] and the three-dimensional JAC3D.[2] These programs provide the analyst with powerful, efficient tools to solve for the nonlinear quasi-static response of solids. In this paper, we describe the development and capabilities of JAC and JAC3D. Following this, we present a selected group of applications relating to the structural analysis of electronic and electromechanical subsystems.

Perspective

Complex problems in structural mechanics can result from the manufacture of electronic and electromechanical subsystems and from the response of subsystems to their use environment. Responses can be characterized as quasi-static—they are time-dependent with constant or negligible inertial effects. They can involve complex geometries, large deformations, nonlinear material behavior, and nonlinear geometric restraints arising from contact surfaces.

Sandia National Laboratories has designed and developed many high-reliability electronic and electromechanical subsystems that can survive severe mechanical, thermal, and radiation environments. In general, evolutionary empirical techniques were initially used in the mechanical design of these subsystems, and the designs were refined by building and testing prototypes. Structural mechanics techniques were applied primarily to evaluate failures and to correct existing designs. The very rapid advancement of structural mechanics techniques has made it possible to assist in optimizing the design before building and testing prototypes, thus minimizing development cost and time. The Engineering Analysis Department at Sandia has developed state-of-the-art finite-element computer programs for specific classes of structural mechanics problems. The programs are expected to be widely used. To facilitate modification of the programs to solve different problems, the codes are structured as modules. For example, the mathematical description of the material behavior (the constitutive model) is contained in separate program subroutines so that new constitutive models can be easily incorporated. The major benefits of these programs are that they provide the structural analyst with accurate, stable, well understood, and computer-efficient analysis tools. For the rest of the problems, commercially available general-purpose finite-element computer programs are used.

Preprocessing (finite-element mesh generation) and postprocessing (results presentation) are separated from the analysis programs. This provides the analyst with a stable set of mesh generation and plotting computer programs.

Solution Techniques

To define the large-deformation, nonlinear static response of a structure subjected to a thermal and/or mechanical load, three sets of equations are required. The first set defines the displacement compatibility (kinematic conditions) of the structure. The second set defines how the material responds (constitutive equations). The third set defines the equations of equilibrium. Combining these equations leads to a set of nonlinear equations which must be solved (or approximated) to determine the nonlinear quasi-static response of the solid body. Mathematically, the problem can be stated as finding the minimum of a functional, $\nabla\Pi(v)$, where the unknowns, v, are the velocity components at the finite element nodal points. Taking the gradient of the functional with respect to the velocities leads to a set of nonlinear equations that can be written as

$$\nabla\Pi(v) = 0 \tag{1}$$

Many current computer programs for structural mechanics rely on a variable-stiffness approach to solve these equations. These programs usually employ either a full or modified Newton-Raphson method. Applying the Newton-Raphson method to the above equation results in a nonzero residual vector R for a given trial solution. Using this residual, which can be interpreted as the vector difference between the internal forces and the external forces, we can write an update for the velocities as

$$\nabla v = -K_t^{-1}R \tag{2}$$

where K_t is the tangent stiffness matrix.

The correction process continues until a suitable norm of the residual satisfies a user-specified tolerance. To a great extent, the success and popularity of the finite-element method are due to this method. Yet, while this method has been (and continues to be) successfully applied, two difficulties arise when the method is extended to large, highly nonlinear problems. First, it is difficult to decide when to reformulate the tangent stiffness matrix to keep the solution from diverging. Second, problem size is often restricted by hardware limitations because, during the solution process, it is necessary to formulate, store, and retrieve the very large stiffness matrix.

Researchers have tried indirect iterative solution techniques on finite-element equations since the early 1960s. Examples are the successive overrelaxation, Gauss-Seidel, and Jacobi methods. Such techniques have been slow to gain acceptance because of the high efficiency of the above-mentioned Newton-Raphson direct solution technique. Indirect iterative solution techniques are now becoming more attractive because of increasing demands to solve large, highly nonlinear static problems and because of advances in computer technology, including vector processing.

After examining various indirect iterative solution techniques, we selected the conjugate gradient method for implementation into a static structural analysis computer program. This technique, chosen for its reliability in obtaining a converged solution, has had great impact in the field of optimization.

Conjugate Gradient Algorithm

For a quasi-static time step (load step), a trial solution of components of the velocity vector is substituted into the set of nonlinear equations [Equation (1)], and a residual vector is obtained:

$$R(v) = \nabla\Pi(v) \tag{3}$$

In the indirect iterative solution procedure, we seek a set of velocity components in which the residual vector is zero or acceptably small. The conjugate gradient method is used to efficiently obtain directions in which to search for the velocity solution. Using a form of the conjugate gradient method obtained by combining a linear preconditioned version[3] and a nonlinear version,[4] we start the iteration process to minimize the functional by assuming a starting vector of velocity components at the nodes of the finite-element mesh, v_j, with j denoting the iteration number. The residual vector, the gradient of the functional, becomes

$$R_j = R(v_j) \tag{4}$$

A preconditioning matrix, M (the diagonal of the linear stiffness matrix), and a generalized gradient vector, Z, is introduced as follows:

$$MZ_j = R_j \tag{5}$$

If $j = 0$, the initial search direction is the negative of the gradient that is the steepest descent direction, P_j,

$$P_0 = -Z_0 = -M^{-1}R_0 \tag{6}$$

Subsequently, for $j > 0$, search directions that are conjugate to the previous direction are chosen as follows:

$$P_j = -Z_j + \beta_j P_{j-1} \tag{7}$$

where β_j has the value

$$\beta_j = \frac{Z_j^T M (Z_j - Z_{j-1})}{Z_{j-1}^T M Z_{j-1}} \tag{8}$$

Equation (8), with the modification using the generalized gradient vector Z is known as the Polak-Ribière algorithm, as discussed in Reference 4. The variables v are then updated by searching for the least value of $\Pi(v)$ from v along the direction P_j. Therefore

$$v_{j+1} = v_j + \alpha_j P_j \tag{9}$$

where α_j is the value that minimizes the function of one variable, known also as a line search. Therefore

$$R(\alpha) = \Pi(v_j + \alpha P_j) \tag{10}$$

If the residual R is not acceptably small, another iteration is begun.

Efficient use of the conjugate gradient method hinges upon the cost of calculating the residual and solving Equation (10) for α.

Calculation of the Residual. For this implementation, we chose the uniform-strain finite element.[5] The element has several advantages: excellent large-deformation response behavior, efficient computation, constant element quantities over the domain of the element, and linear displacement behavior along the edges of the elements.

The residual, which is based on the latest values of v, is a summation of (1) the internal force resulting from the state of stress in each element and the body loads and (2) the external forces resulting from surface tractions. The mean velocity gradients are calculated for each finite element in the deformed configuration. A matrix for finite rotations is found numerically[6] and used to rotate the symmetric part of the velocity gradient (the rate-of-deformation tensor) to a deformed state, called the "unrotated state," having the original orientation. In this unrotated state, constitutive equations are integrated from the beginning of the step to the present state. After the stress is updated, the unrotated stress components are rotated back into the deformed configuration. The internal forces caused by the state of stress in each element can then be determined. The forces resulting from body loads and the forces resulting from tractions on the surface are determined next. Finally, the residual vector is determined by summing each element's contribution to the internal force and subtracting the external forces.

In programming the computer codes, we vectorized all the operations except for the addition of the residual force contributions of each element. The result is an accurate, fast computer code.

Line Search. It is necessary to solve Equation (10) for α. The equations are nonlinear in α, and in optimization it is usual to solve iteratively for α, which can lead to many residual calculations. In mechanics problems involving nonlinear materials, however, only one gradient calculation is needed. This can be approximated by

$$\alpha_j = \frac{Z_j^T M Z_j}{P_j^T R_{P_j}} \qquad (11)$$

The term R_{P_j} represents a residual calculation with the P_j vector substituted for the velocity vector. The material constitutive model is required to calculate a secant modulus for use in calculating R_{P_j}. For a highly nonlinear problem, if a line search is taken in a wrong direction, the conjugate gradient method recovers very slowly. However, use of the secant modulus has proven to give an accurate enough line search to allow effective convergence of the conjugate gradient method.

Constraints. Some mention should be made as to how kinematic constraints are imposed on the finite-element model. Velocity (and therefore displacement) constraints are applied by continuous modification of the residual vector. Updates to the velocities are linearly dependent on the residual vector. Therefore, the appropriate components of the residual vector can be modified after the calculation of the residual, and this modification will be reflected in the update to the velocities in Equation (9). The constraints necessary to simulate different surfaces that are in contact can easily be imposed by modification of the residual vector during each iteration.

Capabilities

The concepts noted above have been incorporated into the structural mechanics computer programs JAC and JAC3D and combined with a variety of ancillary capabilities, resulting in extremely versatile computer programs. The capabilities of both programs are identical except for geometry restrictions. JAC is restricted to either plane strain or axisymmetric geometries; JAC3D is for general three-dimensional solid geometries. These programs, however, often complement one another. For example, in a complicated nonlinear three-dimensional problem, it is often advantageous to perform preliminary two-dimensional calculations to gain understanding of how to model and what to look for in the three-dimensional problem. Having two-dimensional and three-dimensional programs that have identical capabilities and solution techniques allows this to be done efficiently.

Element Birth and Death. The programs have the capability to add elements (element birth) and/or delete elements (element death) at selected times in the solution. This capability has proved to be an important feature, especially in evaluating the residual stresses developed

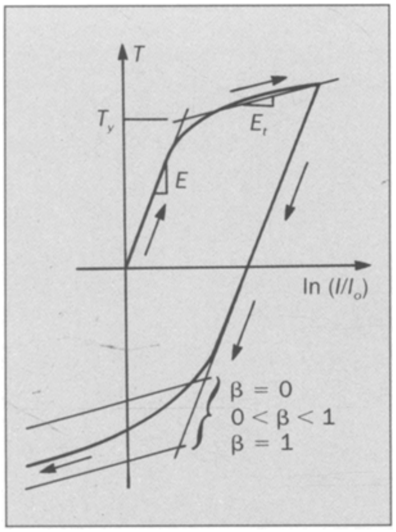


Figure 1. Typical behavior of a ductile metal bar loaded first in uniaxial tension followed by uniaxial compression. The straight line approximation is characterized with an elastic modulus *E*, a yield stress T_y, a strain hardening modulus E_t, and a hardening parameter β. Kinematic hardening is obtained with β = 0, isotropic hardening is obtained with β = 1, and a linear combination of the two is obtained for β between 0 and 1.

during various manufacturing processes. For example, many electronic assemblies are built up in a cascade of soldering steps so that two parts are joined with a high-temperature solder, a third part is joined with a lower-melting-point solder, and so forth. Using the element birth capability, we can realistically model this manufacturing process. In the same manner, changes in residual stress as a result of milling, drilling, or etching can be realistically modeled with the element death capability.

Material Models. At the present time, four nonlinear material constitutive models are incorporated in the programs. For a given problem, any or all of the material models could be used. The models include an isothermal elastic-plastic model, a temperature-dependent elastic-plastic model, a temperature-dependent secondary creep model, and an isothermal finite-strain crushable foam model. The elastic-plastic models are used extensively to describe the response of materials in electronic assemblies. These models are intended to describe classical time-independent plasticity of metallic materials as described in Figure 1 and discussed by Krieg;[7] however, they have been successfully used to describe the behavior of ceramics, rigid polymers, solder at low temperatures, and a host of other materials. The temperature-dependent elastic-plastic model allows the mechanical properties to vary as a function of temperature.

Some materials flow as a function of time (creep) under an applied load or stress. In electronic assemblies, solder is the primary example of a material exhibiting this behavior. A temperature-dependent elastic secondary creep model has been incorporated into the programs. In this model, the strain rate is proportional to stress raised to some power (power law model) which is typically in the range of 3 to 15.

Finally, a material model describing the behavior of materials that crush or compact under application of load has been incorporated in the programs. An example of such a material is low-density polyurethane foam. Because of crushing, classical plasticity models cannot be used to describe the response of these materials. The model includes a pressure-dependent shear failure envelope and arbitrary volumetric plasticity.

The structure is provided so that additional material constitutive models can be easily incorporated.

Kinematic Constraints. The geometric boundary conditions allow for nodal points to be rigidly fixed in space and time or defined to move in a specified time-dependent manner. This capability allows realistic modeling of many quasi-static physical processes. For example, in electronic assemblies, connectors are often required. The mating of a connector pair can be described as press fitting a contact pin into a housing. The requirements are that the contact force be sufficiently high to maintain electrical continuity; however, the stresses in the housing must remain linear elastic so that the connector can be reliably used over and over. Time-dependent boundary conditions applied to the pin in conjunction with a contact surface definition between the pin and housing allow this problem to be modeled easily.

Loads. The programs have the capability to apply a

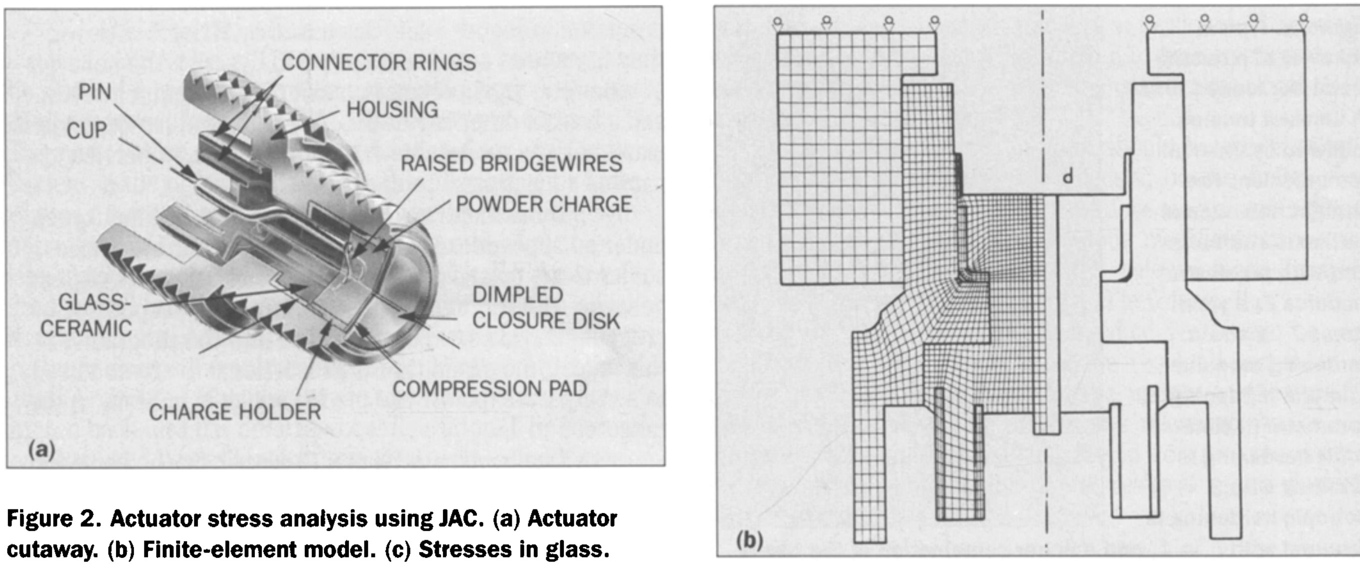


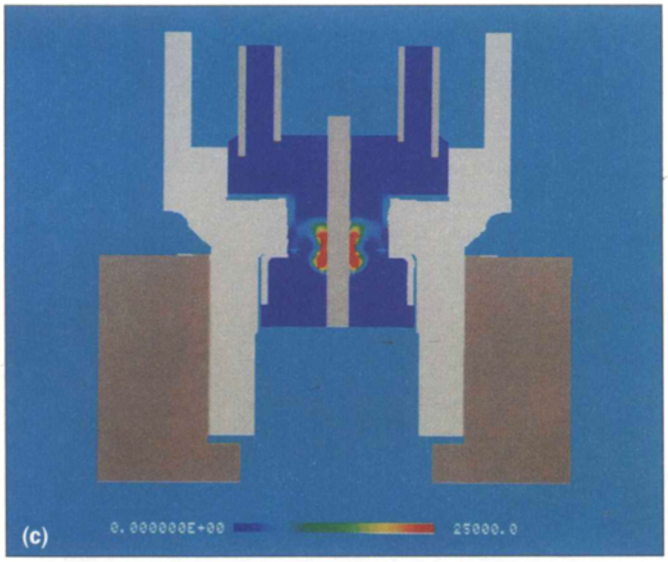
Figure 2. Actuator stress analysis using JAC. (a) Actuator cutaway. (b) Finite-element model. (c) Stresses in glass. (d) Stresses in housing.

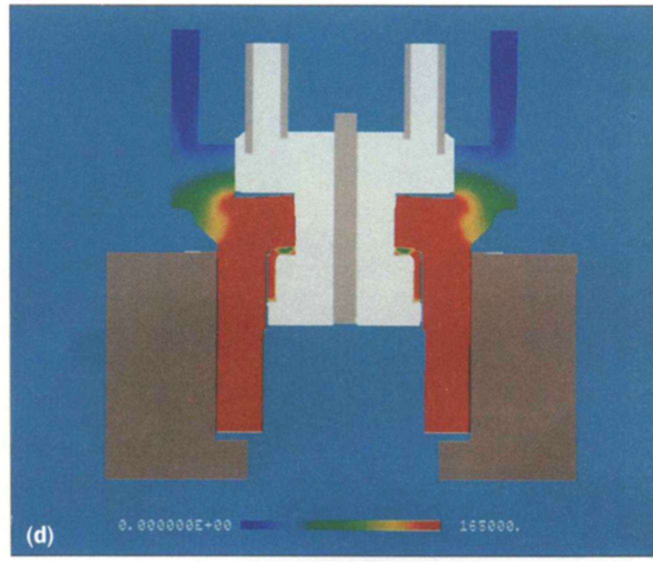
variety of mechanical time-dependent and/or time-constant loads to a model. These loads can be point loads, surface tractions (applied either normal or tangential to the surface), or body forces (arising from uniform acceleration fields). With these definitions, all mechanical loading applications can be modeled.

Thermal Input. The programs have the capability to accept thermal input defining the temperature history of the structure. The temperature history can either be obtained from thermal analysis computer programs or be generated with a user-supplied FORTRAN program. This feature is important for a variety of applications. Residual thermal stresses can be developed during steps in the manufacture of electronics assemblies; examples are soldering and brazing. The stresses are developed because of (1) the difference in the thermal expansion characteristics of the various materials in the assembly or (2) the transient nonuniform temperature history. During use of the electronic assembly, the same problem arises as power is applied or removed.

Contact Surfaces. The programs can also model contacting surfaces. The contact surfaces can be fixed together, sliding without friction, or sliding with friction. They can be allowed to close or open as the solution dictates. This capability allows many physical processes to be realistically modeled—connector insertion, for example. The "fixed" contact surface has proved to be very useful for grading element size, especially for the three-dimensional problem. This allows parts of the structure to be modeled in a very fine manner to get the required resolution. Yet the remaining part, which is required to obtain the global response, can be modeled in a coarse manner. These parts are joined by one or more fixed contact surfaces.

Restart. Finally, a capability to restart the solution is also incorporated. The restart can be used to change many of the problem parameters, thus allowing realistic physical processes to be modeled easily. For instance, stresses and deformations are generally developed in an assembly during manufacture. Use environments impose additional stresses and deformations on the assembly. With the restart capability, an analysis of the manufacturing environment has to be completed only once. Various sub-





sequent-use environments can then be evaluated.

Applications

Four applications are presented to show the versatility and capability of JAC and JAC3D. In addition to these applications in electromechanical and electronics packaging, the programs have been applied extensively in other fields of applied mechanics such as geomechanics.

For our examples, the material behavior is approximated by either the isothermal elastic-plastic or the temperature-dependent elastic-plastic constitutive model.

High-Strength Actuator Header. Mechanical functions such as valve closure are often driven by the pressure developed by burning a pyrotechnic powder. Pyrotechnic-driven actuators are electrically ignited with a bridge-wire. The structure surrounding the powder, called the header, must not only be of sufficient strength to contain the pressure but must also have features to allow for the electrical connection. Ceramics or glass-to-metal seals are often used in the design of the headers to provide the required electrical insulation. JAC was successfully applied to three aspects of a design process for optimizing a high-strength header for a pyrotechnically driven actuator.[8] This design process was a joint effort of experts from several disciplines including design engineering, material science, testing, manufacturing, and structural analysis.

After material selection, JAC was applied to evaluate the residual stresses resulting from a glass-to-metal seal manufacturing process in which a high-strength S-glass ceramic[9] insulator was joined to a nickel-steel housing. Results from these finite-element analyses were used to identify manufacturable header designs with a minimum residual stress state. JAC was then used to obtain the response of the header to pyrotechnic burn.

The results provided (1) realistic upper bounds on the pressure containment ability of various preliminary header designs, and (2) a quick, inexpensive way of strengthening and refining the designs. Because testing of the headers was difficult and sometimes destructive, analytical results were also used to interpret test results and identify failure modes. Figure 2a shows the final design configuration of the high strength actuator. Figure 2b is an axisymmetric finite element idealization of the header.

Analytical results indicate that even though the glass-ceramic insulator cracks extensively (Figure 2c) and the nickel-steel housing yields extensively (Figure 2d), the header can withstand internal pressures in excess of 150,000 lb/in^2. The predicted failure modes and levels compared favorably with observed test results.

Resistor Lead Failure and Packaging Redesign. Electronic packages are commonly encapsulated to allow them to survive severe shock and vibration environments and to isolate them from high voltage. Encapsulant materials range from low-density polyurethane or polystyrene foams to filled rigid epoxies. The encapsulant materials have, in general, much higher thermal expansion than the materials of the electronic assemblies. The difference in thermal expansion can result in severe stresses being developed in components and electrical interconnections during normal operational thermal cycling.

During thermal cycle testing, the leads of the resistor shown in Figure 3a failed, causing an open electrical circuit in a small percentage of electronics packages. Because of the complex geometry, we used JAC3D to perform a three-dimensional analysis. The finite-element mesh (Figure 3b) had approximately 7000 nodal points and 5800 three-dimensional elements. The model included a fixed contact surface that allowed the use of (1) a fine mesh around the resistors to achieve the required resolution and (2) a coarse mesh in other areas to minimize model size. The analysis indicated that resistor leads bend severely during thermal cycling (Figure 3c), causing enough stress to result in failure. The calculations showed that the observed failure of the resistor leads was caused by the encapsulant.

Because JAC3D is a highly efficient, vectorized code, a number of parametric calculations could be economically performed to guide the redesign of the package.

Ceramic Vacuum Tube Analysis. Figure 4a shows a ceramic high-voltage vacuum tube mounted in an assembly package. Subsequently, the vacuum tube is encapsulated in a filled rigid epoxy. During thermal cycling tests, a small percentage of the ceramic tubes cracked (Figure 4b). A series of two-dimensional axisymmetric finite-element calculations provided critical information on the magnitude of residual stresses in the ceramic vacuum tube resulting from its manufacture and yielded substantial insight as to how to improve packaging of the tube in the assembly package.

Three-dimensional calculations were required to explain the observed mode of failure. The three-dimensional finite-element idealization is shown in Figure 4c. The stress resulting from thermal cycling was calcu-

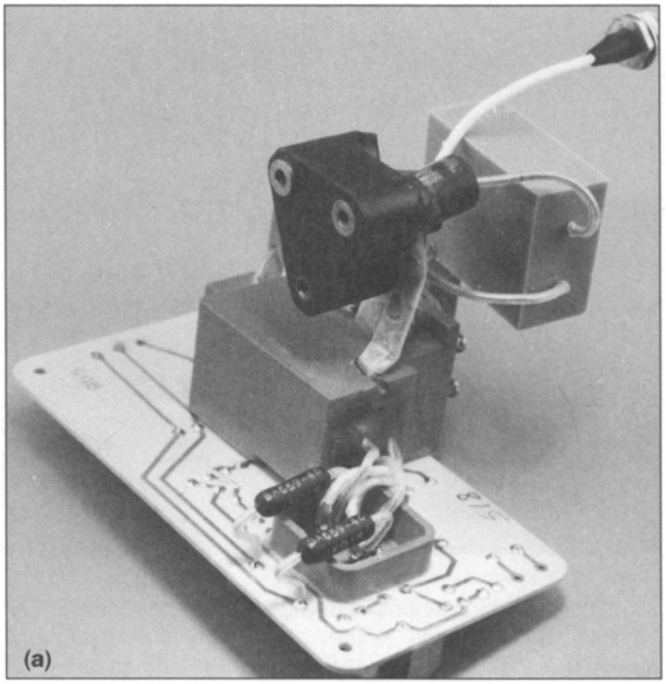
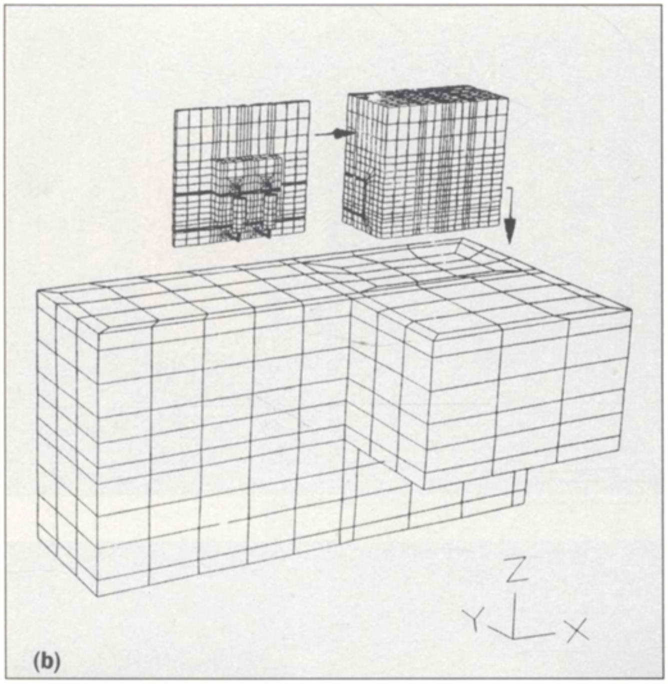
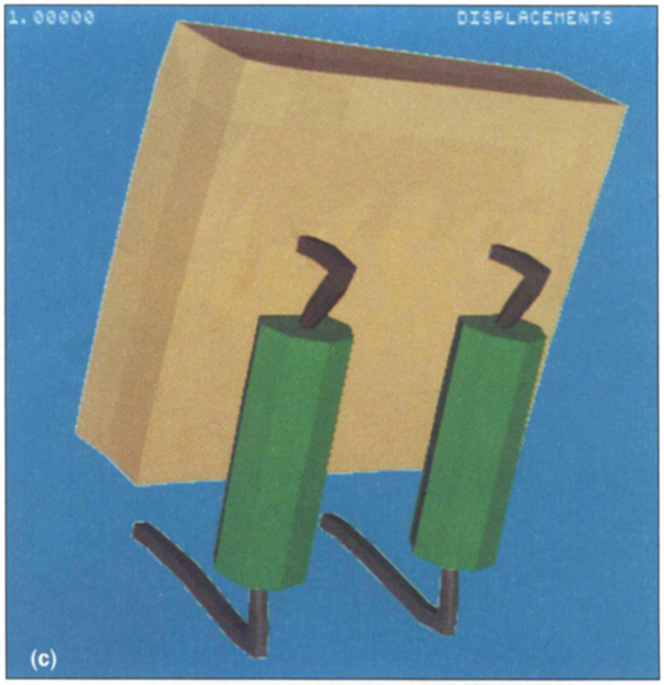


Figure 3. Analysis of resistor lead failure using JAC3D. (a) Component module with resistors. (b) Three-dimensional finite-element idealization. (c) Deformed shape of resistors at low temperature.



(b)



(c)

lated by using JAC3D. The calculated maximum tensile principal stresses in the ceramic were not symmetric around the tube. The location of the peak value of tensile stress coincided with the location of the failure. Parametric calculations were performed to evaluate proposed design changes. Results indicated that the stresses in the ceramic tube could be reduced to acceptable levels through a relatively minor design change consisting of symmetrically locating the tube in the encapsulation material.

Microcircuit Surface Mount Solder Study. The mounting of microcircuit chip carriers to various printed wiring boards, such as ceramic substrates or multilayer boards, raises many design and reliability questions. In commercial applications, microcircuit carrier solder joints are subjected to repeated thermal excursions because of equipment power being turned on and off. However, in military applications, not only are solder joints subjected to thermal excursions, but the assemblies must also survive severe shock and vibration environments.

In the past, these questions have been evaluated by building and testing prototypes. While testing of the final design is necessary, it is costly and time-consuming in the design phase. Additionally, a design that survives the environmental testing may not be the most satisfactory design.

If realistic mechanical structural analyses of these assemblies can be performed, the effects of various manufacturing and design parameters can be evaluated and used to arrive at the most satisfactory design. Because of the complicated three-dimensional geometry of assemblies and the nonlinear material response of solder, analysis has been limited. Analysis has also been limited by the fact that very fine resolution is required in the rather small sol-

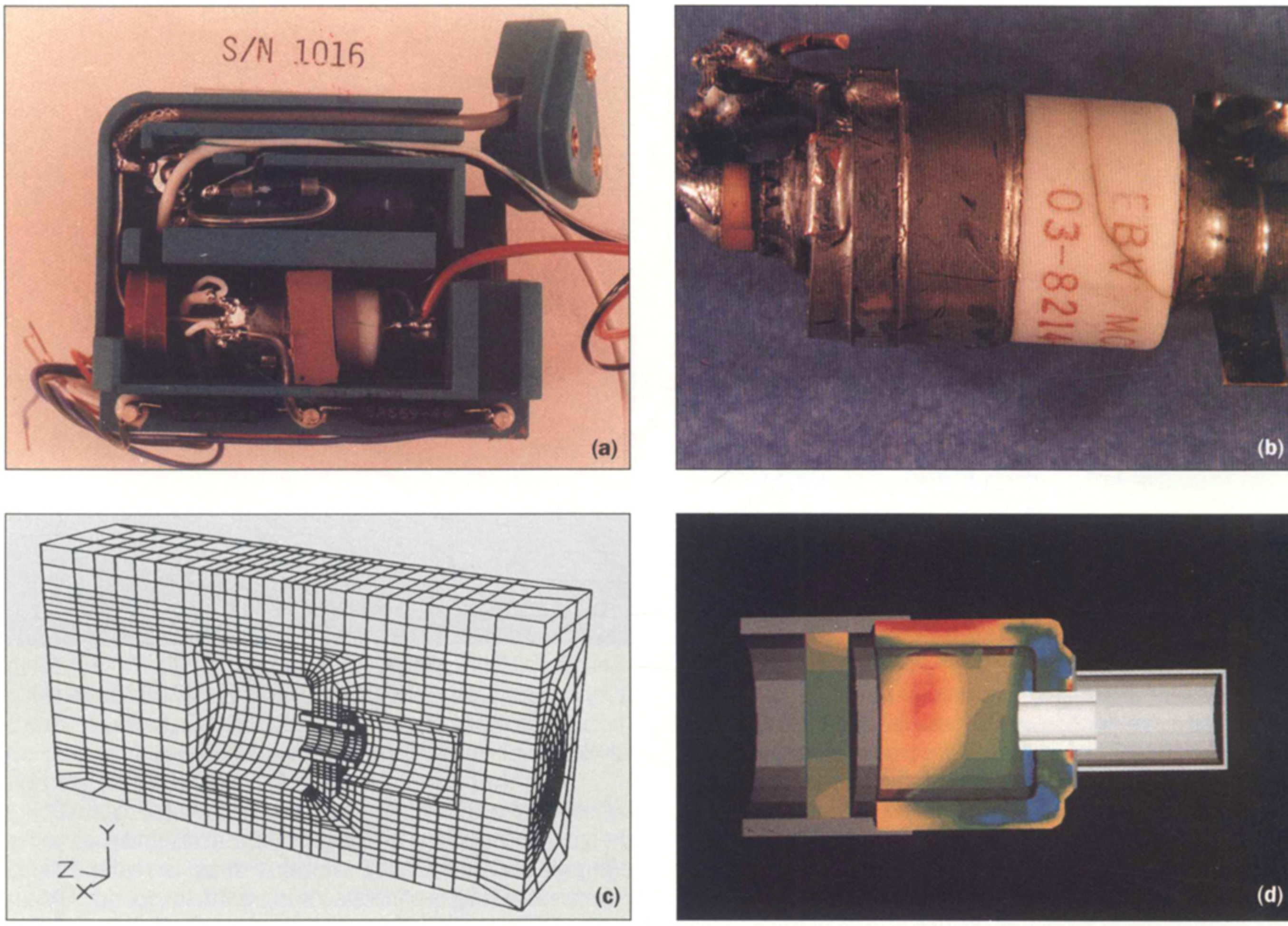


Figure 4. Analysis of cracked ceramic vacuum tube using JAC3D. (a) Component package without encapsulation. (b) Cracked tube. (c) Finite-element idealization. (d) Calculated stress in tube.

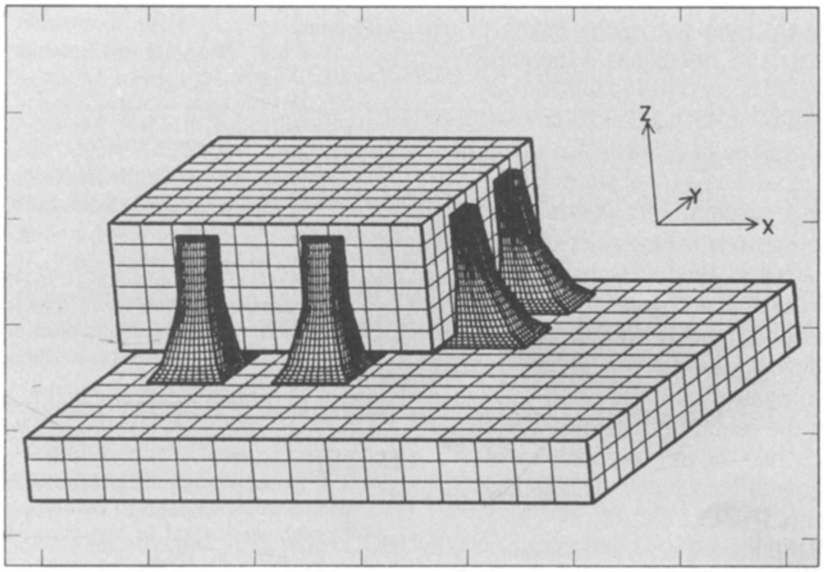


Figure 5. Package solder study—three-dimensional finite-element idealization.

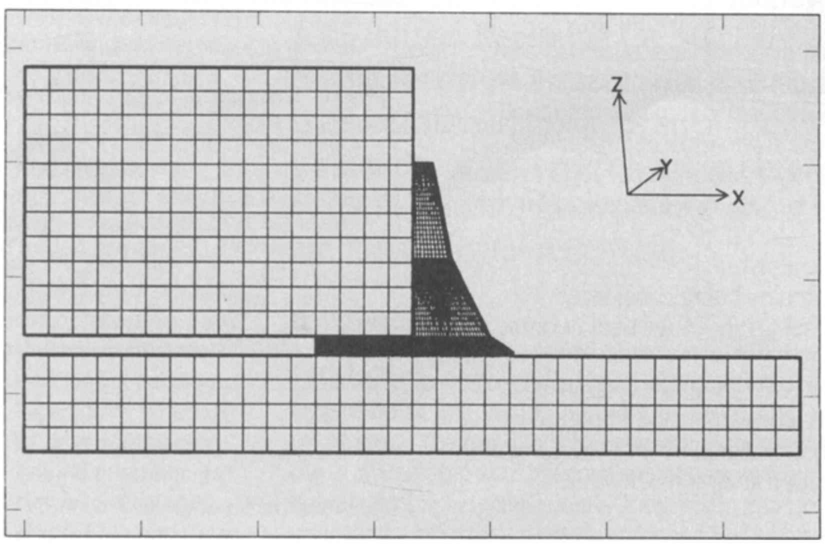


Figure 6. Package solder study—two-dimensional finite-element study.

der post; however, because the loads are often carried simultaneously along several paths, modeling of the entire redundant structural geometry is required to obtain the loading into the solder joints.

A study has been initiated to evaluate the effect of solder post geometry on the residual stress state developed in a ceramic chip carrier mounted on a ceramic substrate as a result of cooling down after soldering. Of particular interest are the stresses tending to pull the circuit metallization from the ceramic substrate. A break in the metallization would cause an open circuit.

The three-dimensional finite-element idealization of a chip carrier is shown in Figure 5. By use of symmetry and fixed contact surfaces, the required resolution can be obtained in the solder post while the entire geometry of the surface-mounted component is included. This model has approximately 14,000 nodal points and 11,000 elements. A calculation to determine the residual stresses created in the assembly by initial soldering required approximately 300 seconds on a Cray computer.

A two-dimensional plane strain model was also developed, as shown in Figure 6. In this particular problem, results from the two-dimensional model essentially agreed with results from the three-dimensional model. Therefore the two-dimensional model was used to evaluate various geometric variations. The two-dimensional model would not be adequate if:

- Thermal loading is not uniform.
- Loading other than thermal is being considered.
- A printed wiring board substrate other than ceramic is being considered.

Although the study is in its early stages, preliminary results suggest that the stresses that tend to pull the metallization from the substrate depend on solder volume rather than any single geometric dimension.

Acknowledgment

This work performed at Sandia National Laboratories is supported by the U.S. Department of Energy under Contract Number DE-AC04-76DP00789.

References

1. J. H. Biffle, "JAC—A Two-Dimensional Finite Element Computer Program for the Non-Linear Quasistatic Response of Solids with the Conjugate Gradient Method," Sandia National Laboratories report SAND81-0998, Albuquerque, New Mexico, 1984.
2. J. H. Biffle, "JAC3D—A Three-Dimensional Finite Element Computer Program for the Nonlinear Quasi-Static Response of Solids with the Conjugate Gradient Method," Sandia National Laboratories report SAND-87-1305, Albuquerque, New Mexico, to be published in February 1988.
3. P. Consus, G. H. Golub, and D. P. O'Leary, "A Generalized Conjugate Gradient Method for the Numerical Solution of Elliptic Partial Differential Equations," *Sparce Matrix Computations*, J. R. Bunch and D. J. Rose, eds., Academic Press, New York, 1965, pp. 307-332.
4. M. J. D. Powell, "Restart Procedures for the Conjugate Gradient Method," *Mathematical Programming*, Vol. 12, pp. 242-254, 1977.
5. D. P. Flanagan and T. Belytschko, "A Uniform Strain Hexahedron and Quadrilateral with Orthogonal Hourglass Control," *International Journal for Numerical Methods in Engineering*, Vol. 17, pp. 679-706, 1982.
6. D. P. Flanagan and L. M. Taylor, "An Accurate Numerical Algorithm for Stress Integration with Finite Rotations," *Computer Methods in Applied Mechanics and Engineering*, Vol. 62, 1987, pp. 305-320.
7. R. D. Krieg and S. W. Key, "Implementation of a Time Independent Plasticity Theory into Structural Computer Programs," *Constitutive Equations in Viscoplasticity: Computational and Engineering Aspects*, J. A. Stricklin and K. J. Saczalski, eds., ASME publication AMD-Vol. 20, 1976, pp. 125-138; also an invited lecture at the 1976 ASME Winter Annual Meeting, New York.
8. S. N. Burchett, "Optimizing Header Strength Utilizing Finite Element Analysis," *Proceedings*, Eleventh International Pyrotechnics Seminar, July 1986, IIT Research Institute, Chicago, pp. 645-666.
9. H. L. McCollister and S. T. Reed, United States Patent 4,414,283.

(Manuscript received September 9, 1987)