NASA Technical Memorandum 102723

Error Detection and Control for Nonlinear Shell Analysis

Susan L. McCleary
Lockheed Engineering and Sciences Company
Hampton, Virginia

Norman F. Knight, Jr.
NASA Langley Research Center
Hampton, Virginia

(NASA-TM-102723) ERROR DETECTION AND CONTROL FOR NONLINEAR SHELL ANALYSIS (NASA) 12 p CSCL 20K

N90-28377

Unclas

G3/39 0305304

August 1990

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23665-5225

ERROR DETECTION AND CONTROL FOR NONLINEAR SHELL ANALYSIS

Susan L. McCleary
Lockheed Engineering and Sciences Company
Norman F. Knight, Jr.
NASA Langley Research Center
MS 244, NASA Langley Research Center, Hampton, VA 23665-5225 USA

SUMMARY

A problem–adaptive solution procedure for improving the reliability of finite element solutions to geometrically nonlinear shell–type problems is presented. The strategy incorporates automatic error detection and control and includes an iterative procedure which utilizes the solution at one load step from one finite element model to obtain an equivalent solution at the same load step on a more refined model. Representative nonlinear shell problems are solved.

INTRODUCTION

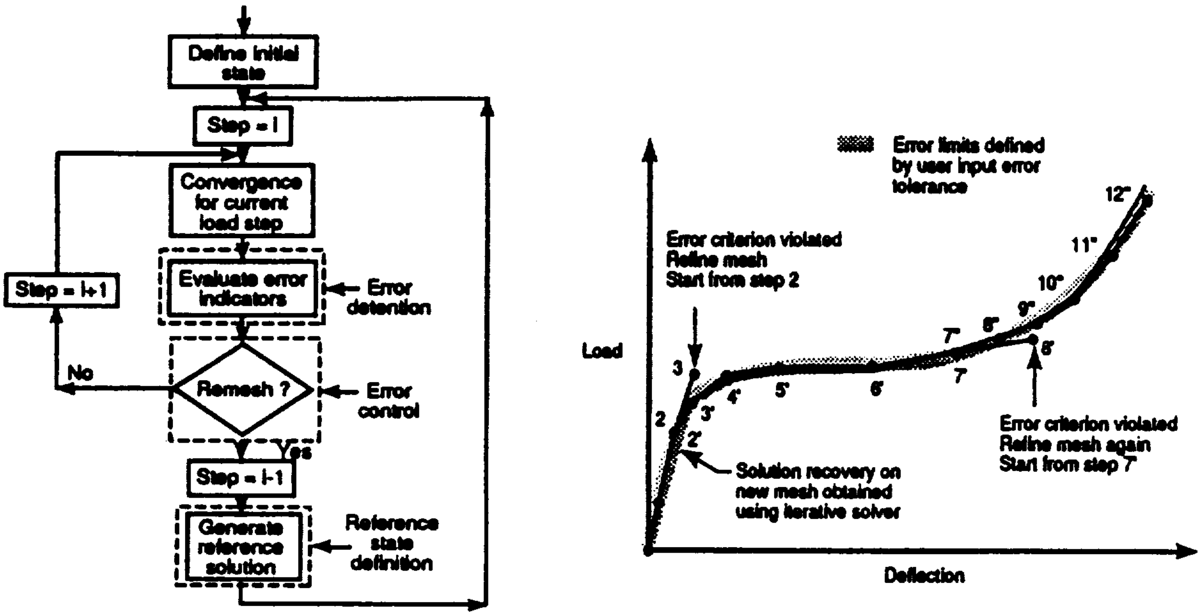
Much of the research in adaptive finite element structural analysis has centered on the development of techniques for use in linear analysis. However, the need for an adaptive strategy is even more important in nonlinear analysis where a given finite element model may perform adequately for a certain range of loading (*e.g.*, for the first several load steps) and become grossly inadequate for another range of loading (*e.g.*, for the last several load steps). Recent work at the NASA Langley Research Center has focused on the development of an adaptive nonlinear analysis procedure for shell structures (McCleary [1]). This adaptive analysis procedure integrates three primary components into the nonlinear solution procedure: an automatic error detection strategy, an automatic error control strategy, and a reference state definition technique. Each of these components are described herein, and the use of the procedure is demonstrated on two geometrically nonlinear shell problems.

THE ADAPTIVE ANALYSIS PROCEDURE

The primary components of the adaptive nonlinear solution procedure are identified in the algorithm flowchart of Figure 1(a). The use of this algorithm to predict a nonlinear response curve is illustrated in Figure 1(b). The analyst defines an error tolerance which sets boundaries on the deviation of the calculated response from the converged response. At some load step, the initial finite element discretization may predict a response which violates the prescribed error tolerance (*e.g.*, see step 3 in Figure 1). At this point, a new finite element model is generated and the solution procedure backs up to the previous load step (see steps 2 and 2′ in the figure). Once an equivalent solution has been obtained at the previous load step for the new finite element mesh, the adaptive procedure proceeds until the response again violates the prescribed error tolerance (*e.g.*, step 8′ on the figure).

Error Detection

Many adaptive mesh refinement strategies employ a two tiered *a posteriori* error detection technique. In this technique, a global error estimate is computed as the summation of the local error



(a) (b)

Figure 1. Adaptive nonlinear solution procedure

indicators, and is examined to determine if refinement of the finite element mesh is necessary. When required, local error indicators are used to identify the elements which need to be modified. Generally, the global error estimate is an accurate measure of the error in the finite element solution. An alternate technique employs an error indicator on an element-by-element basis to provide qualitative rather than quantitative information about the solution. In this approach, local error indicators are used only to guide mesh refinement. An estimate of the exact global error is not calculated.

This second approach has been adopted in the present study and may be viewed as element-independent in that element-specific information (*e.g.*, element shape functions) is not accessed. The error indicators are based on the use of stress resultants rather than the generalized nodal displacements. A smoothed stress resultant field, obtained by nodal averaging, is used to approximate the "exact" solution. As a consequence of using stress resultants, the error detection procedure is indirectly uncoupled from the large deflection/large rotation formulation used to account for geometric nonlinearity. Specific knowledge of the form of the strain displacement relations (*e.g.*, linear or nonlinear) or the nonlinear formulation (*e.g.*, total Lagrange, corotational) is not required. As such, the procedure is applicable to the geometrically nonlinear analysis of plate and shell structures.

Two error indicators have been employed in this study. The first is an adaptation of the energy norm of the error as proposed by Zienkiewicz and Zhu [2]. The second indicator uses the discontinuities in the finite element stress resultant field in the form of a standard deviation in stress resultants.

The error detection strategy developed by Zienkiewicz and Zhu [2] for membrane problems and later extended to plate bending problems by Dow and Byrd [3], is based on the energy norm of the error. These element error indicators are defined herein by the energy norm of the error in stress resultants within each element. Namely

$$\|e_k\|_E^2 = \int_{\Omega_k^e} (S_k^* - S_k^i)^T \overline{C}_k^{-1} (S_k^* - S_k^i) d\Omega_e = \int_{\Omega_k^e} \Delta S_k^T \overline{C}_k^{-1} \Delta S_k d\Omega_e \tag{1}$$

where, for the k^{th} element, S_k^* is a vector of smoothed stress resultants (continuous across interelement boundaries), S_k^i is a vector of discrete stress resultants recovered from the finite element solution, and $\overline{C}_k$ is the constitutive matrix that relates stress resultants to reference surface strain components.

The energy norm of the error as defined in equation (1) is an integrated quantity and therefore requires either knowledge of the element shape functions or an element–independent generalization. The distribution of the difference between the smoothed and discrete values of the stress resultants, ΔS_k, over each element is assumed to be described by Lagrange shape functions. The application of the energy norm of the error as an element error indicator has at least two shortcomings. First, the evaluation of the integral in equation (1) may become computationally expensive for large problems. Second, the error is in terms of the error in energy and thus provides only indirect information about the quality of the secondary solution (*i.e.*, stress resultants).

A new indicator has been developed in an attempt to minimize the computational effort and to provide more direct information about the quality of the stress resultant field. From statistics, the coefficient of variation of the stress resultants, V^{SR}, is defined as an error indicator. In this work, V^{SR} is defined at each node j in terms of the standard deviation in the stress resultants and the maximum absolute value of the smooth stress resultants (denoted as $|\dot{S}_\alpha^*|_{max}$) as

$$V_{\alpha_j}^{SR} = \frac{100}{|S_\alpha^*|_{max}} \sqrt{\frac{\sum_{i=1}^{n_e} (S_{\alpha_j}^{''} - S_{\alpha_j}^*)^2}{n_e - 1}} \tag{2}$$

The symbol n_e is the number of elements connected to node j, and S_α is a single stress resultant component. It should be noted that $V_{\alpha_j}^{SR}$ is defined for each stress resultant component. The elemental component value, $V_{\alpha_k}^{SR}$, is taken to be the maximum nodal component value for a given element. The maximum coefficient of variation is used in order to ensure a conservative estimate of the error within an element as measured by $V_{\alpha_k}^{SR}$.

<u>Error Control</u>

The development of an error control strategy requires the definition of refinement indicators and the development of an automatic mesh refinement strategy. Refinement indicators are used to identify regions of the finite element mesh requiring an adjustment to their discretization and are formulated based on the previously defined error indicators. The mesh refinement strategy, based on h–extension, determines precisely how these identified regions are refined.

The refinement indicator based on the energy norm of the error is defined as in reference [2]. Elements needing refinement are identified by using the local refinement indicator ξ_k defined by

$$\xi_k = \frac{\|e_k\|_E}{e_m} \tag{3}$$

The quantity e_m is the projected elemental error assuming the current mesh is an optimally refined mesh (*i.e.*, the error uniformly distributed over the elements). The user–prescribed error tolerance is defined as a percentage of the global strain energy.

The refinement indicator based on the coefficient of variation of stress resultants is formulated by requiring that the analyst specify a maximum acceptable standard deviation in stress resultants as a fraction of the absolute maximum smoothed stress resultant. A single element refinement indicator is formed using the Euclidean norm, *i.e.*,

$$\xi_k = \frac{1}{\bar{\eta}}\sqrt{\sum_{\alpha}\xi^2_{\alpha_k}} = \frac{1}{\bar{\eta}}\sqrt{\sum_{\alpha}\left(\frac{V^{SR}_{\alpha_k}}{100}\right)^2} \tag{4}$$

where the error tolerance $\bar{\eta}$ is the fraction of the global maximum smoothed stress resultant. The use of an Euclidean norm in the definition of ξ_k ensures a conservative estimate of the refinement requirements for a given element.

When the value of ξ_k exceeds unity in a given element, the element is marked for refinement. A value of ξ_k less than unity for a patch of elements indicates that the patch should be less refined or undergo "fusion" (*i.e.*, combining the elements in a patch of elements into a patch containing fewer elements). Zienkiewicz and Zhu [2] use an element sizing parameter, h/h_k (equal to $1/\xi_k$ for 4-node elements and $1/\sqrt{\xi_k}$ for 9-node elements), to predict multiple levels of refinement. The current implementation contains no provisions for either multi-level refinement or the fusion of elements.

An automatic mesh refinement strategy may take several forms and need not necessarily be fully adaptive. A fully adaptive strategy is able to refine selectively. Automatic mesh refinement encompasses any mesh refinement technique which can be performed automatically by the structural analysis software. However, automatic mesh refinement which is not fully adaptive will generally tend to over-refine the finite element model.

The automatic mesh refinement strategy employed in this study is denoted as quasi-uniform mesh refinement since uniform refinement may be carried out within individual, predefined regions of the finite element model. Quasi-uniform refinement operates best on finite element models which are easily parameterized so that it may be performed automatically by changing only the few parameters which define the finite element model. Within the framework of a general-purpose finite element code, quasi-uniform mesh refinement is perhaps the most expedient approach in that it may be performed at a high level (*e.g.*, using a command language), it does not require special data structures or software, and perhaps more importantly, it always uses the precise definition of the structural geometry in the remeshing process.

<u>Reference State Definition</u>

After a new finite element mesh has been defined, the nonlinear solution from the previous mesh needs to be defined for the new mesh. This solution on the new mesh corresponds to the reference state for restarting the nonlinear solution algorithm. Several factors must be considered in deciding the manner in which the reference state is defined. The reference state should be some intermediate solution and not the initial state (*i.e.*, the solution procedure should not have to start from the first load step of the first finite element model). Computing the new reference state should not be computationally expensive, should not further complicate data structures, and should not require access to element-specific information (such as shape functions). With these requirements in mind, a new approach has been developed in which interpolation of displacement data is not employed. The reference state is instead defined using an iterative solution recovery procedure [1].

```
form applied displacement vector
j = 1
do for j < max_d_iters
   u^j_{i-1} = u^{j-1}_{i-1}
   form tangent stiffness matrix based on u^j_{i-1}
   solve K^j_{i-1} u^j_{i-1} = f_{i-1} using PCG method
   calculate error:
    e = || u^j_{i-1} - u^{j-1}_{i-1} || / || u^j_{i-1} ||
   if e < tol exit
   j = j + 1
enddo
```

Figure 2. Iterative solution recovery.

The strategy for determining a consistent solution for a refined mesh is summarized in Figure 2. This strategy uses the nonlinear solution at the previous load step (step $i-1$) from the previous finite element model, to generate an applied displacement vector for the solution at the same load step on the new finite element model. Only those nodes which are new to the finite element model are left unconstrained during the iteration cycle. An initial estimate of the solution vector is formed by initializing the displacements at the new nodes to zero. While these initial displacements may be assigned values other than zero, doing so has not proven to alter significantly the convergence characteristics of the iterative solution recovery procedure. Once formed, the initial estimate is used to generate a tangent stiffness matrix and serves as an initial guess for a preconditioned conjugate gradient (PCG) iterative solver. The solution u^j_{i-1} is compared to the solution at iteration $j-1$ (*i.e.*, u^{j-1}_{i-1}) and the solution recovery error is measured by

$$e^j_u = \sqrt{\frac{\Delta u^T_{i-1} \Delta u_{i-1}}{u^{j\,T}_{i-1} u^j_{i-1}}} \tag{5}$$

where $\Delta u_{i-1} = u^j_{i-1} - u^{j-1}_{i-1}$. If the error, e^j_u is less than the specified error tolerance, a converged solution at step $i-1$ has been achieved. If the error is greater than the specified tolerance, a new tangent stiffness matrix is formed based on the j^{th} displacement solution. The procedure continues until a converged solution for load step $i-1$ on the new finite element mesh is obtained. The algorithm then proceeds to load step i and obtains an improved solution at load step i on the new finite element mesh.

This technique is particularly effective, and only efficient, when an iterative equation solver (*e.g.*, PCG technique) is used. In this case, the solution u^{j-1}_{i-1} is used as an initial solution for the iterative solver at iteration j (*i.e.*, as an initial guess for u^j_{i-1}), thereby increasing the rate of convergence for the solver. While direct solvers may be used, they require a complete factorization of the tangent stiffness matrix at each iteration and will take roughly the same amount of time for each solution iteration. An iterative solver will generally require an ever decreasing number of internal iterations as the overall solution recovery process converges.

NUMERICAL RESULTS

The adaptive procedure described herein has been developed using the framework of the CSM Testbed [4]. Control of the analysis procedure is implemented using the command language

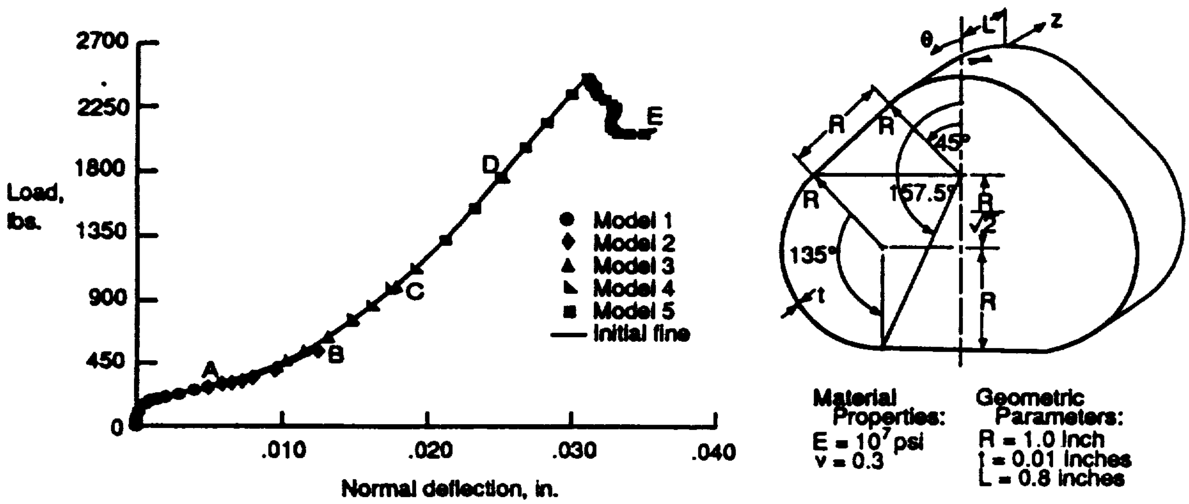


Figure 3. Pear–shaped cylinder: geometry, properties, and response.

capability of the CSM Testbed. Moreover, a stand–alone Fortran processor has been developed for evaluating the various error and refinement indicators. A total Lagrangian formulation is used for the nonlinear problem description, and the solution strategy incorporates the Riks/Crisfield arc–length approach within a modified Newton–Raphson procedure to solve the system of nonlinear algebraic equations (see [1] for details). The refinement indicator, ζ_k, displayed in the following sections is analogous to the element sizing-parameter previously discussed and is defined as $\zeta_k = 1/\xi_k$ for 4–node elements and as $\zeta_k = 1/\sqrt{\xi_k}$ for 9–node elements.

In subsequent sections, the results of two nonlinear shell analyses are presented. Both problems involve structures which exhibit complex nonlinear behavior. The first problem, the collapse of a pear–shaped cylinder, demonstrates the use of the adaptive procedure on an isotropic, geometrically regular (rectangular elements are used throughout the domain) problem. Correlation with previously published results is shown. The second problem, the buckling of a cylindrical panel with a hole, demonstrates the use of the adaptive procedure on a composite, geometrically irregular (skewed elements are required to model the central circular hole) problem. Correlations with test data and previously published analytical results are shown.

Pear–Shaped Cylinder

The pear-shaped cylinder shown in Figure 3 has been adopted by many researchers and the behavior of this cylinder subject to a uniform end-shortening investigated (see Hartung and Ball [5]). The cylinder response becomes nonlinear at low values of applied end-shortening, and the normal deflections of the flat portions of the cylinder increase rapidly with increases in loading. The symmetry exhibited by the structure allows the analysis to be performed on one fourth of the cylinder.

Quasi–uniform mesh refinement has been used for error control. The finite element mesh has been divided into four groups of elements. Element group 1 contains the elements in the 45° arc at the top of the cylinder, element group 2 contains the elements in the flat segment between the two curved segments, element group 3 contains the elements in the 135° arc, and element group 4 contains the elements in the bottom flat segment of the cylinder. Circumferential refinement

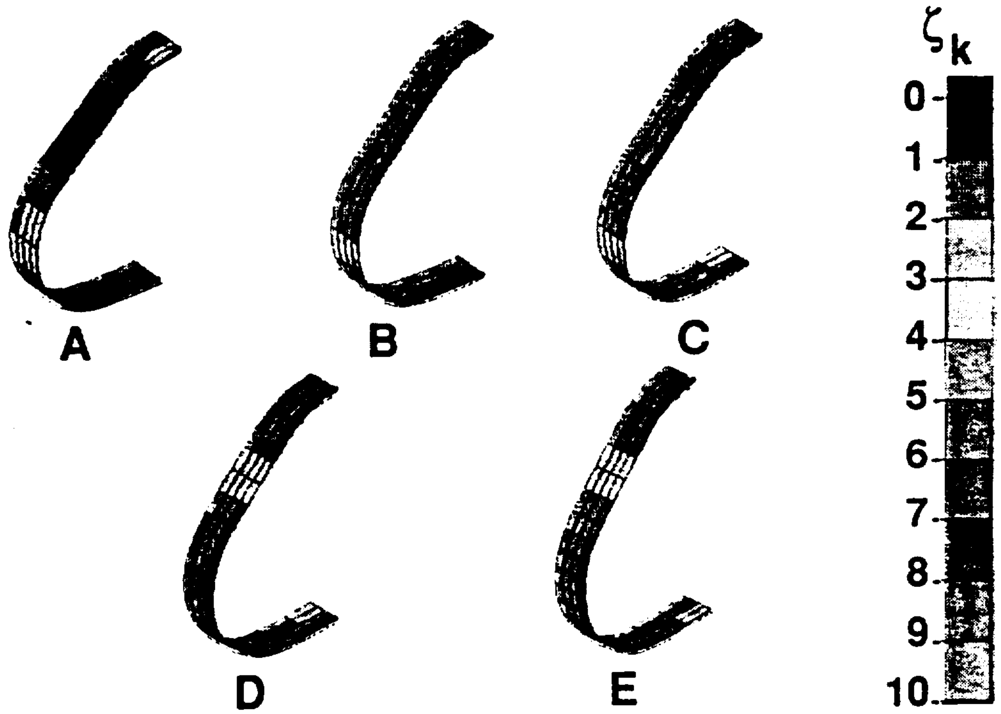


Figure 4. Refinement indicators, ζ_k, for finite element models 1 through 5. Models A–E correspond to points A–E on Figure 3.

within any individual group of elements was unrestricted. Whenever all four element groups required refinement simultaneously, refinement through the depth of the cylinder was carried out in a uniform fashion (*i.e.*, for all elements in all groups). Command language procedures were used to direct the mesh refinement based on the original cylinder geometry.

The nonlinear response of a series of five finite element models is also shown in Figure 3. As shown in the figure, once the solution starts to deviate from the converged solution (603 nodes, 132 9-node Assumed Natural–coordinate Strain (9ANS) elements [6]), mesh refinement occurs. The iterative solution recovery algorithm then defines the reference state for the restart of the nonlinear solution procedure. The collapse load is 2471 lbs., within 5% of the collapse load reported by Hartung and Ball [5].

For this series of models, refinement is based on the energy norm of the error with a 10% error tolerance; each model is composed of 9ANS elements. The refinement indicators ζ_k for the set of finite element models are shown in Figure 4. Initially, refinement was required only within the flat portions of the finite element model where significant nonlinear behavior was present. At higher values of load, the curved segments of the cylinder also needed refinement since these segments began to exhibit large normal deflections. Similar results are obtained using the coefficient of variation of stress resultants with an error tolerance of 10%.

Composite Cylindrical Panel

The postbuckling response of axially compressed composite cylindrical panels with holes has been a subject of research for several years (*e.g.*, Knight and Starnes [7]). This problem is characterized by large local deformations in the neighborhood of the hole which cause ply delaminations to occur in the postbuckling range. The panel analyzed herein is loaded with a uniform end–shortening

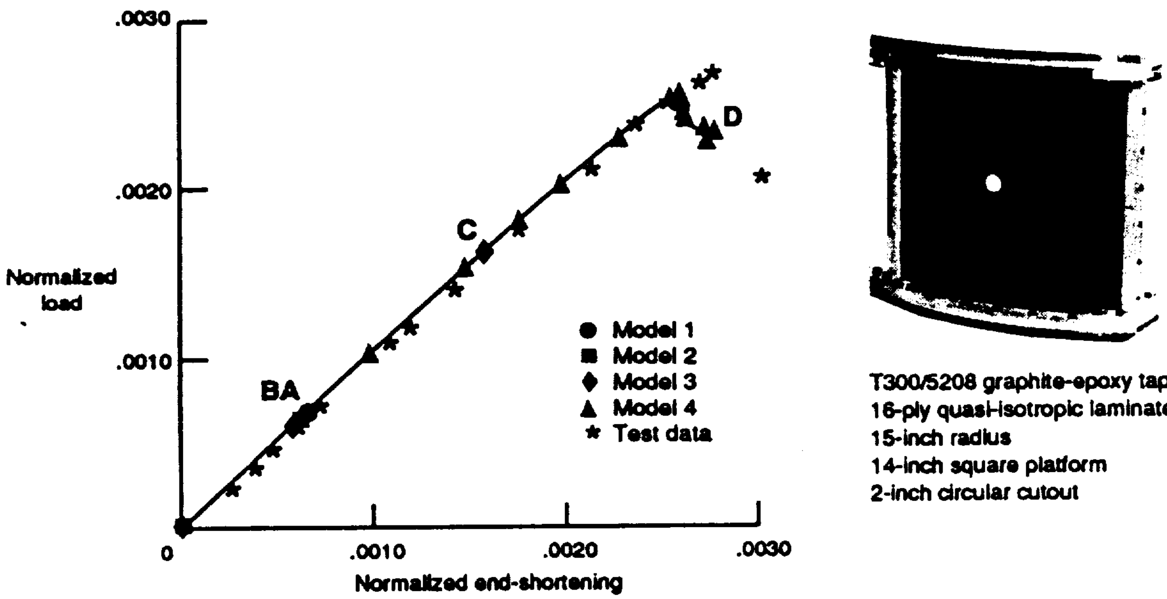


Figure 5. Composite cylindrical panel: geometry, properties, and response

and is referred to as panel CP8 in reference [7]. The panel geometry and material and section properties are shown in Figure 5.

To obtain a correct solution, the curvature of this panel must be preserved throughout the mesh refinement process. The exact panel geometry is difficult to represent because of the presence of a cylindrical surface with a curved interior boundary. In general, the finite element mesh will require many distorted elements in order to adequately represent the panel geometry. A uniform mesh refinement strategy was used to accomplish mesh refinement. Once any single element was flagged for refinement, uniform mesh refinement was carried out by alternately adding spokes of nodes and rings of elements. A command language procedure was used to direct this mesh refinement.

The analyses presented in this section were halted soon after buckling occurred due to the inadequacy of the material model after buckling (as reflected in the experimental results). The converged solution shown in Figure 5 (which plots the normalized load versus end–shortening response) was obtained using a finite element mesh with 1600 nodes and 384 9ANS elements. While the predicted buckling load is below the experimental data, it is consistent with the predicted buckling load given in reference [7]. In addition, the effects of geometric imperfection were not included in the analysis which may account for some of the differences between the experimental and analytical results.

A series of four finite element meshes was generated using a 15% error tolerance on the coefficient of variation of stress resultants. The model used 4-node Assumed Natural–coordinate Strain (4ANS) elements [6]. The resulting response curve is also shown in Figure 5. Again the solution recovery algorithm had no difficulty in transitioning from one finite element mesh to another. Three finite element meshes were automatically generated before the error indicators computed at the first load step indicated that an acceptable solution had been obtained. The refinement indicators ζ_i for this series of finite element meshes are shown in Figure 6 in a planform view. As the model is refined, the elements requiring refinement move along the diagonals of the panel toward the edge

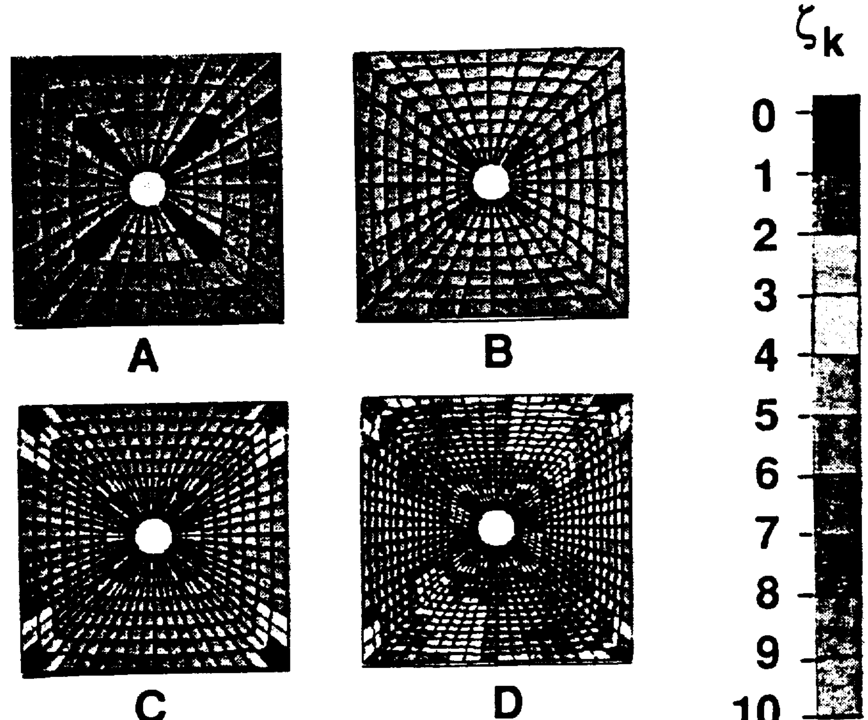


Figure 6. Refinement indicators, ζ_k, for finite element models 1 through 4. Models A–D correspond to points A–D on Figure 5.

of the hole where large gradients occur.

<u>Computational Effort</u>

One of the goals of the work described thus far is to show that the adaptive nonlinear solution strategy presented herein can be efficient. Satisfying this goal requires that the computational time required for performing a nonlinear analysis using the adaptive procedure as outlined should not substantially exceed the computational time required for a nonlinear analysis performed using a single refined finite element model. Timing data for both the pear–shaped cylinder and the composite cylindrical panel are summarized in Table 1. For each case, the single refined finite element model was selected to be the final finite element model generated by the adaptive solution procedure. This single refined model was defined only after the adaptive analysis was performed. In general, multiple models and analyses would have been required to obtain a reliable solution.

In the case of the pear–shaped cylinder, the analysis performed using the adaptive procedure required only approximately 9% more CPU time than the analysis performed using the single refined finite element model. In the case of the composite cylindrical panel, the analysis performed using the adaptive procedure required approximately 19% more CPU time than the analysis performed using a single refined model. This difference in the additional percentage of CPU time required is attributable to the fact that all of the mesh refinement occurs within the first two load steps of the analysis of the composite panel. While the analysis performed using the single refined model required less CPU time for both problems, the reliability of the solutions can only be assessed by a complete re–solution on a different mesh. In both cases, the total CPU time required for computing two complete nonlinear solutions would be greater than the CPU time required to perform the single adaptive analysis.

The results shown in the table are very encouraging in that the adaptive analysis is at least

Table 1. Timing Data for Adaptive and Single Nonlinear Analyses

	Pear–Shaped Cylinder		Composite Cylindrical Panel	
Function	Adaptive Analysis Time†	Single Refined Model Time†	Adaptive Analysis Time†	Single Refined Model Time†
Model Generation	100.6	4.2	81.7	34.5
Form and Factor Stiffness	2561.2	2497.5	3254.9	3062.0
Matrix Operations	967.0	949.7	584.5	575.0
Error Calculation	117.0	–	435.3	–
Total	3746.1	3452.8	4358.6	3672.5

† In CPU seconds. Calculations performed on a Convex C220 minisupercomputer.

competitive with the analysis performed on the single refined finite element model. If a fully adaptive (*i.e.*, selective) mesh refinement strategy was employed, at least two improvements in the efficiency of the procedure would be possible. First, the model generation phase of the analysis would be more efficient. The current implementation requires a complete regeneration of each model rather than an incremental change in the previous model. Second, improvements could be made in the iterative solution recovery procedure by partitioning the tangent stiffness matrix and only reforming the elemental matrices for the "new" elements. The current implementation requires the complete regeneration of the tangent stiffness matrix for each solution iteration.

CONCLUDING REMARKS

A problem–adaptive nonlinear solution procedure has been described. This procedure incorporates automatic error detection and control into a modified Newton–Raphson nonlinear solution strategy. A technique for defining a reference state, called iterative solution recovery, has been developed. This technique has been shown to be an effective means of making the transition from one finite element discretization to another at a given load level. The use of two error indicators, the energy norm of the error and the coefficient of variation of stress resultants, as effective guides for mesh refinement, has been demonstrated. Two geometrically nonlinear shell problems have been solved and good correlation with published results has been presented.

REFERENCES

1. McCleary, S.L.: *An Adaptive Nonlinear Analysis Procedure for Plates and Shells.* M.S. Thesis, George Washington University, Hampton, VA, 1990.
2. Zienkiewicz, O.C.; and Zhu, J.Z.: A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis. *International Journal for Numerical Methods in Engineering,* Vol. 24, 1987, pp. 337–357.
3. Dow, J.O.; and Byrd, D.E.: An Error Estimation Procedure for Plate Bending Elements. AIAA Paper No. 88–2318.
4. Knight, Jr., N.F.; Gillian, R.E.; McCleary, S.L.; Lotts, C.G.; Poole, E.L.; Overman, A.L.; and Macy, S.C.: *CSM Testbed Development and Large–Scale Structural Applications.* NASA TM-4072, April, 1989.
5. Hartung, R.F.; and Ball, R.E.: *A Comparison of Several Computer Solutions to Three Structural Shell Analysis Problems.* AFFDL-TR-73-15, U.S. Air Force, April, 1973.
6. Stanley, G.M.: *Continuum–Based Shell Elements.* Ph.D. Dissertation, Stanford University, Stanford, CA, 1985.
7. Knight, Jr., N.F.; and Starnes, Jr., J.H.: Postbuckling Behavior of Axially Compressed Graphite-Epoxy Cylindrical Panels with Circular Holes. *ASME Journal of Pressure Vessel Technology,* Vol. 107, 1985, pp. 394–402.

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
NASA TM-102723		

4. Title and Subtitle	5. Report Date
Error Detection and Control for Nonlinear Shell Analysis	August 1990
	6. Performing Organization Code
7. Author(s)	8. Performing Organization Report No.
Susan L. McCleary Norman F. Knight, Jr.	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 Technical Memorandum
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, DC 20546-0001	14. Sponsoring Agency Code

15. Supplementary Notes

Susan L. McCleary, Lockheed Engineering and Sciences Company, Hampton, Virginia

Norman F. Knight, Jr., Formerly NASA Langley Research Center, Hampton, Virginia 23665
Currently Clemson University, Clemson, South Carolina

Presented at the Sixth World Congress on Finite Element Methods, October 1-5, 1990, Banff, Alberta, Canada

16. Abstract

A problem-adaptive solution procedure for improving the reliability of finite element solutions to geometrically nonlinear shell-type problems is presented. The strategy incorporates automatic error detection and control and includes an iterative procedure which utilizes the solution at one load step from one finite element model to obtain an equivalent solution at the same load step on a more refined model. Representative nonlinear shell problems are solved.

17. Key Words (Suggested by Authors(s))	18. Distribution Statement
Error detection and control Nonlinear shell analysis	Unclassified—Unlimited Subject Category 39

19. Security Classif.(of this report)	20. Security Classif.(of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	11	A03

NASA FORM 1626 OCT 86

For sale by the National Technical Information Service, Springfield, Virginia 22161-2171