Conf-9206253--1

PNL-SA--21006

DE92 019138

GLOBAL-LOCAL FINITE ELEMENT ANALYSIS OF COMPOSITE STRUCTURES

J. E. Deibler

June 1992

Presented at the
Swanson Analysis Systems Technology Conference
June 8-11, 1992
Pittsburg, Pennsylvania

Work supported by
the U.S. Army
Armament Research and Development Center
under a Related Services Agreement
with the U.S. Department of Energy
under Contract DE-AC06-76RLO 1830

Pacific Northwest Laboratory
Richland, Washington 99352

GLOBAL-LOCAL FINITE ELEMENT ANALYSIS OF COMPOSITE STRUCTURES

John E. Deibler
Pacific Northwest Laboratory [a)]

ABSTRACT

The development of layered finite elements has facilitated analysis of laminated composite structures. However, the analysis of a structure containing both isotropic and composite materials remains a difficult problem. A methodology has been developed to conduct a "global-local" finite element analysis. A "global" analysis of the entire structure is conducted at the appropriate loads with the composite portions replaced with an orthotropic material of equivalent material properties. A "local" layered composite analysis is then conducted on the region of interest. The displacement results from the "global" analysis are used as loads to the "local" analysis. The laminate stresses and strains can then be examined and failure criteria evaluated.

INTRODUCTION

The increasing usage of composite materials in structural applications has led to the continued development of advanced layered or composite elements in the ANSYS (1989) finite element program. A triangular laminated shell element (STIF53) was available at least as far back as Revision 3.0. The 100 layer STIF99 shell element was released in Revision 4.3 and had the capability of evaluating failure criteria. Revision 4.4 included the 100 layer STIF46 solid element which includes through thickness effects.

The initial composite structures to be fabricated were thin laminates. The assumption of plane stress conditions simplified analysis methods. Advances in composite fabrication techniques have made possible the production of thick composite structures. The plane stress assumption was no longer valid and analysts requested elements with through thickness capabilities. The release of the STIF46 element in Revision 4.4 provided a means for analyzing thick layered solids or shells.

Structures comprised totally of composite materials are uncommon at this time. Conventional materials seem invariably to be a part of the structure whether for temperature, moisture, corrosion, fabrication, cost or other reasons. This combination of isotropic and laminated materials is particularly perplexing to the analyst if the isotropic portion would normally be modelled with 2-dimensional elements, either plane stress, plane strain, or axisymmetric. The typical approach has been to replace the composite portion of the structure with an orthotropic material of equivalent stiffness so that stresses in the isotropic portions of the structure can be evaluated. An independent laminate

a) Pacific Northwest Laboratory is operated for the U.S. Department of Energy by Battelle Memorial Institute under Contract DE-AC06-76RLO 1830

analysis code is then applied to the composite section using loads determined from the finite element analysis.

The concept of substituting a material of equivalent stiffness for a laminated composite structure is not new in solid mechanics. Pagano and Soni (1983) used variational principles to derive a global-local model which was capable of determining accurate global displacements and local lamina stresses. Hyer (1987) used an equivalent orthotropic stiffness technique to evaluate the displacements of a thick, laminated cylinder. Kumar and Weerth (1991) have documented a procedure using ANSYS with a separate post-processor to examine interlaminar stresses.

This paper describes a method that was developed to conduct the entire global-local analysis within the ANSYS program. The composite portion of the model is replaced by an equivalent orthotropic material. The displacement results of this global finite element analysis are applied to a local three-dimensional composite model. The Cut Boundary Interpolation method located in the ANSYS AUX1 utility is used to accurately transfer the displacements from the global model to the local model. Lamina stresses can then be evaluated and failure criteria considered.

METHODOLOGY

The first step is the determination of the equivalent orthotropic properties that represent the composite laminate. This equivalent stiffness material is used in the global analysis of the entire structure. This is accomplished with the STIF46 element.

The STIF46 element is an 8-node solid element designed to model thick layered shells or solids. This element allows up to 100 different layers to be modeled in one element. The material, orientation, and thickness of each layer is defined with the element real constants. An alternative modeling approach is to stack multiple elements with each element representing one or more layers. Both methods are used during the course of this procedure.

A unit dimension STIF46 element is constructed and the desired composite layup is specified with the element real constants. Table 4.46.1 of the ANSYS User's Manual describes the required information. Correct use of coordinate systems as pertaining to material directions is of utmost importance throughout this entire procedure. The examples given here are consistent with an axisymmetric analysis of a filament wound structure (i.e., ANSYS global Y-axis is the axis of revolution).

ANSYS scales the input lamina thicknesses to the thickness of the element as defined by the nodes. This allows the use of a unit dimension element. The laminate can be described with actual layer thicknesses without being concerned that the element has the necessary thickness.

The key to the use of ANSYS to determine equivalent material properties is in setting KEYOPT(2) = 5 for the STIF46 element. This forces ANSYS to print the material property matrices after integrating through the thickness of the element. These are labeled MEMBRANE, INTERACTION, BENDING, THIRD-ORDER, and FOURTH-ORDER matrix. The first three are the familiar A, B and D stiffness orientation be selected and/or graph plots (PLPATH) be used. The typically wide variation in stress levels between lamina results in a loss of detail if contour plots of the entire model are made. Figure 2 shows a set of results obtained by a PLPATH command of elements selected by common material and orientation. If the analyst is confident in the failure criterion used, color contour plots of the entire model would provide a ready means for identifying lamina that exceed the failure criterion.

RESULTS

This procedure was applied extensively to the design of an externally pressurized cylindrical structure. Failure predictions from the analysis correlated well with the failure modes observed during experimental testing.

ACKNOWLEDGEMENTS

The author wishes to express his appreciation to the U.S. Army Armament Research, Development and Engineering Center for sponsoring this work and permitting its presentation.

REFERENCES

DeSalvo, G.J., and Gorman, R.W. ANSYS Engineering Analysis System User's Manual. Rev 4.3, 4.4, Swanson Analysis Systems, Inc., 1987, 1989.

Agarwal, B.D, and Broutman, L.J. Analysis and Performance of Fiber Composites. John Wiley and Sons, New York, 1980.

Kelley, F.S. "Mesh Requirements for the Analysis of a Stress Concentration by the Specified Boundary Displacement Method", Swanson Analysis Systems, Inc.

Kumar, V., and Weerth, D.E. "Finite Element Analysis of Thick Composite Plate Structures", Composites Design, Manufacture, and Application. Stanford University, 1991.

Hyer, M.W. "Response of Thick Laminated Cylinders to External Pressure", Department of Mechanical Engineering, University of Maryland, Technical Report 87-4, 1987.

Pagano, N.J., and Soni, S.R. "Global-Local Laminate Variational Model", Int. J. Solids Structures 19, 207 1983.

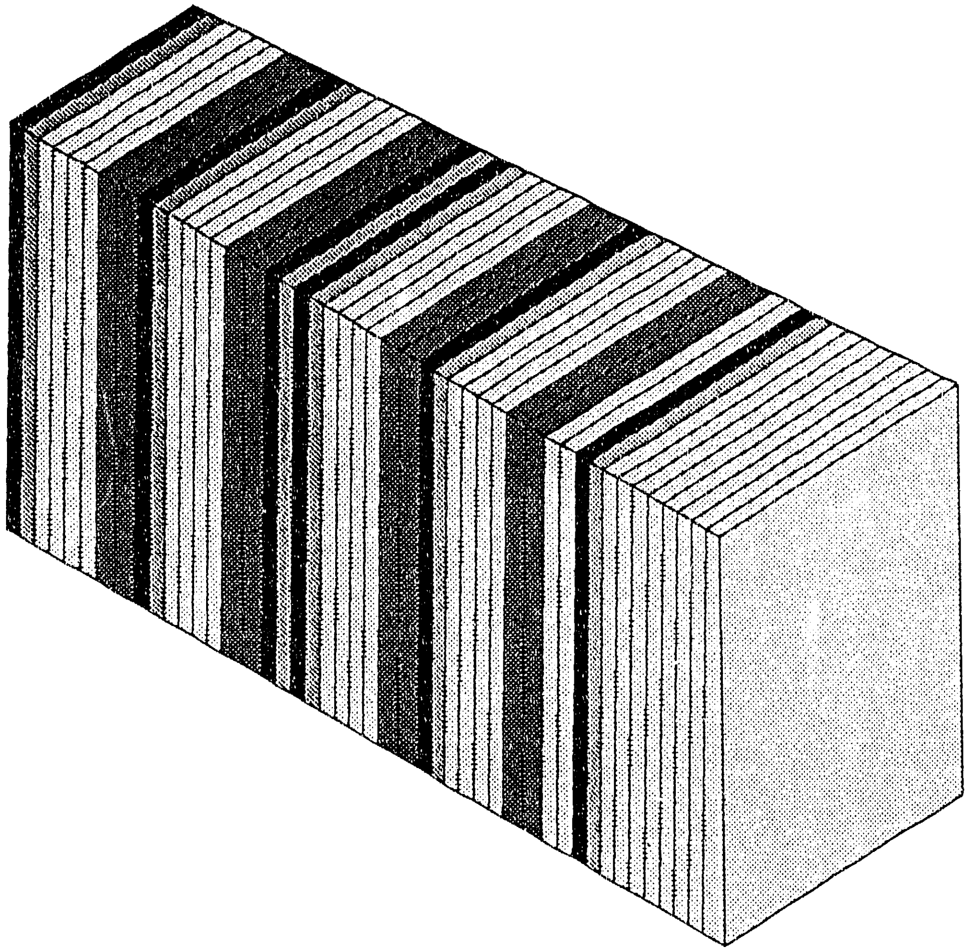
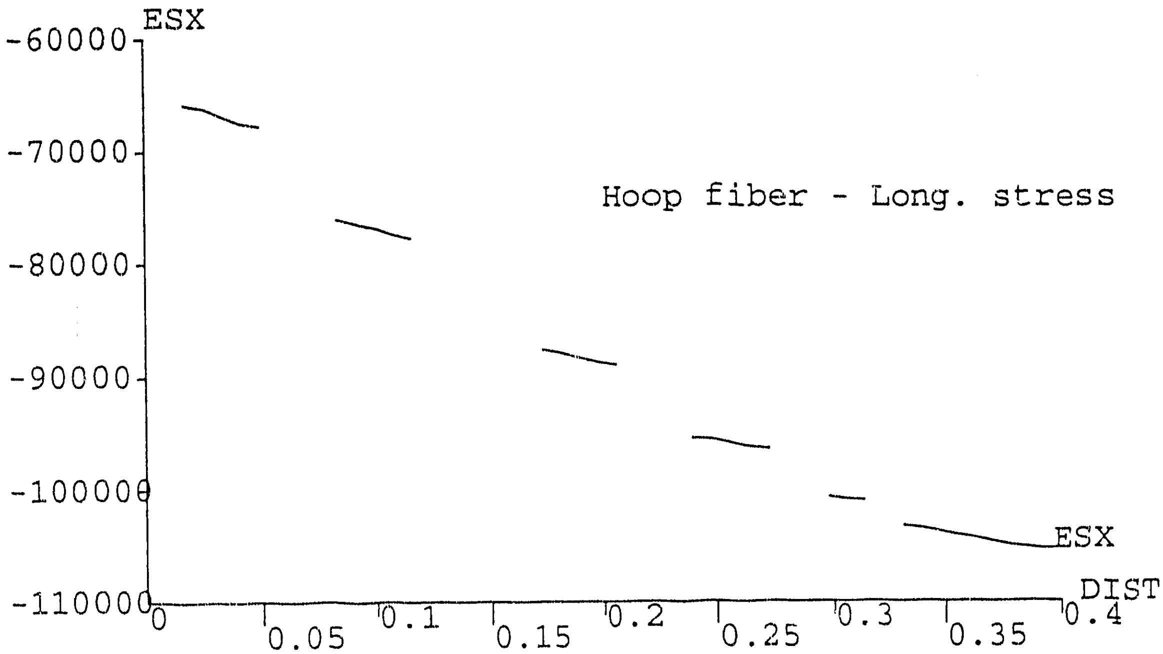


Figure 1 STIF46 Local Model



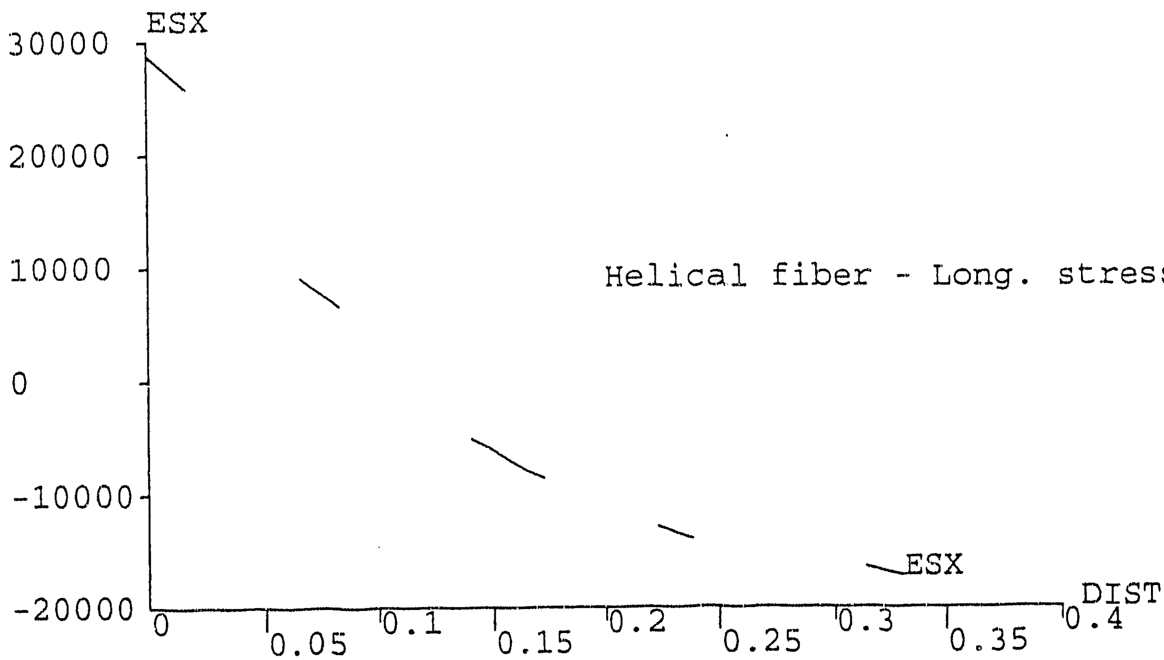


Figure 2 Lamina Stress Results

END

DATE FILMED

9 / 28 / 92