MODELING MATERIAL FAILURE WITH A VECTORIZED ROUTINE

S. M. CRAMER
AND
J. R. GOODMAN

DEPARTMENT OF CIVIL ENGINEERING
COLORADO STATE UNIVERSITY

FORT COLLINS, COLORADO

MODELING MATERIAL FAILURE WITH A VECTORIZED ROUTINE

S. M. Cramer
Research Associate
Dept. of Civil Engineering
Colorado State University
Ft. Collins, Colorado

J. R. Goodman
Professor
Dept. of Civil Engineering
Colorado State University
Ft. Collins, Colorado

ABSTRACT

The computational aspects of modeling material failure in structural wood members are presented with particular reference to vector processing aspects. Wood members are considered to be highly orthotropic, inhomogeneous, and discontinuous due to the complex microstructure of wood material and the presence of natural growth characteristics such as knots, cracks and cross grain in wood members. The simulation of strength behavior of wood members is accomplished through the use of a special purpose finite element/fracture mechanics routine, program STARW (STrength Analysis Routine for Wood). Program STARW employs quadratic finite elements combined with singular crack tip elements in a finite element mesh which accounts for the complexities inherent in wood structural members. The need to use a highly refined finite element mesh to adequately model material behavior, results in the formulation of thousands of simultaneous equations which must be generated and solved repeatedly to model the nonlinear failure process which occurs. The availability of the CYBER 205 at Colorado State University has made implementation of program STARW at the level described not only possible, but also relatively economical. Vector processing techniques are employed in mesh generation, stiffness matrix formation, simultaneous equation solution, and material failure calculations. The paper addresses these techniques along with the time and effort requirements needed to convert existing finite element code to a vectorized version. Comparisons in execution time between vectorized and nonvectorized routines are provided.

INTRODUCTION

Accurate knowledge of the strength of a structural member is essential information to the design engineer concerned with structural safety and efficient material use. A means to predict material strength is necessary, since all materials exhibit some variability in strength and it is not feasible to physically test every structural member to determine its load carrying capacity. The sophistication of strength prediction models have generally advanced, not only with the discovery and refinement of new computational methods, but also with the increase in computer capabilities which enable efficient application of the new methods.

In the case of wood structural members, the current strength prediction method is a highly approximate procedure based on empirical concepts from the 1930's. This results in a strength prediction that is relatively uncertain. The current strength prediction procedure is based on the results of physical tests because until now it has not been possible to mathematically model wood member failure and rationally predict strength. The most obvious difficulties; orthotropic material properties, the presence of knots and associated grain deviations, and the presence of cracks from seasoning and partial material failure, can now be successfully modeled with program STARW (STrength Analysis Routine for Wood) (2).

The nature of the nonlinear failure modeling process, presents a computational problem of such a large magnitude that it can not be efficiently accomplished on computers that do not have the capacity of a CYBER 205. Program STARW represents a case where modest effort in invoking vector processing syntax has not only made implementation of the program possible, but has also resulted in a relatively economical solution.

AN OVERVIEW ON MATHEMATICALLY MODELING WOOD MEMBER FAILURE WITH STARW

Program STARW uses two-dimensional orthotropic finite elements to model behavior in the longitudinal-transverse plane of a loaded wood member. Tensile load is applied in the longitudinal direction as shown in Fig. 1.

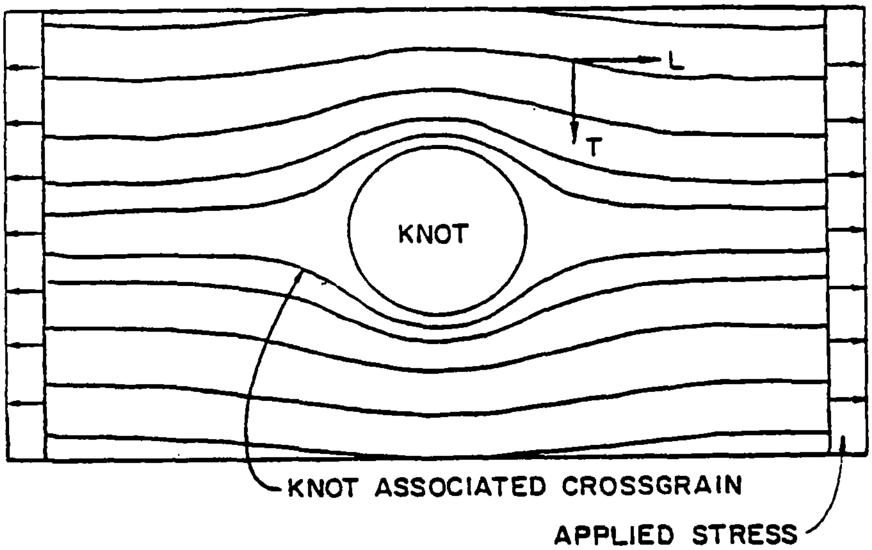

Figure 1. Loaded Wood Structural Member (Longitudinal-Transverse Plane)

A knot in a structural specimen of wood creates localized grain deviation as indicated in Fig. 1. This grain deviation has an extremely important effect on stress distributions at locations near the knot (3). An iterative procedure to locate mesh coordinates corresponding to the grain deviation around a knot is employed in program STARW. This procedure relates distortion of wood grain around a knot to streamlines of laminar fluid flow around an elliptical object and has therefore been named the "flow-grain analogy" (4).

Utilizing the flow-grain analogy, a representative finite element mesh is automatically constructed of eight node quadrilateral elements, six node triangular elements, and eight node singular elements. Since tangential elastic stiffness of wood may be as little as 1/20 of the longitudinal elastic stiffness, all three types of finite elements are required to model different elastic material behavior in the longitudinal and tangential directions. Appropriate elastic stiffness values for each element are automatically assigned.

Singular elements are used to model material behavior around the tip of cracks that form as the load on the member is increased. These elements were developed using theory from linear elastic fracture mechanics (1). Experimental investigations have indicated that cracks in structural lumber will usually form and propagate along a grain line. Thus, cracks are modeled by program STARW by "unzipping" the finite element mesh along the material separation and placing the singular elements around the crack tip. A resulting finite element mesh is shown in Fig. 2. The "unzipping" process and placement of the singular elements are performed automatically upon cue by the user when the appropriate failure conditions are indicated in the program output.

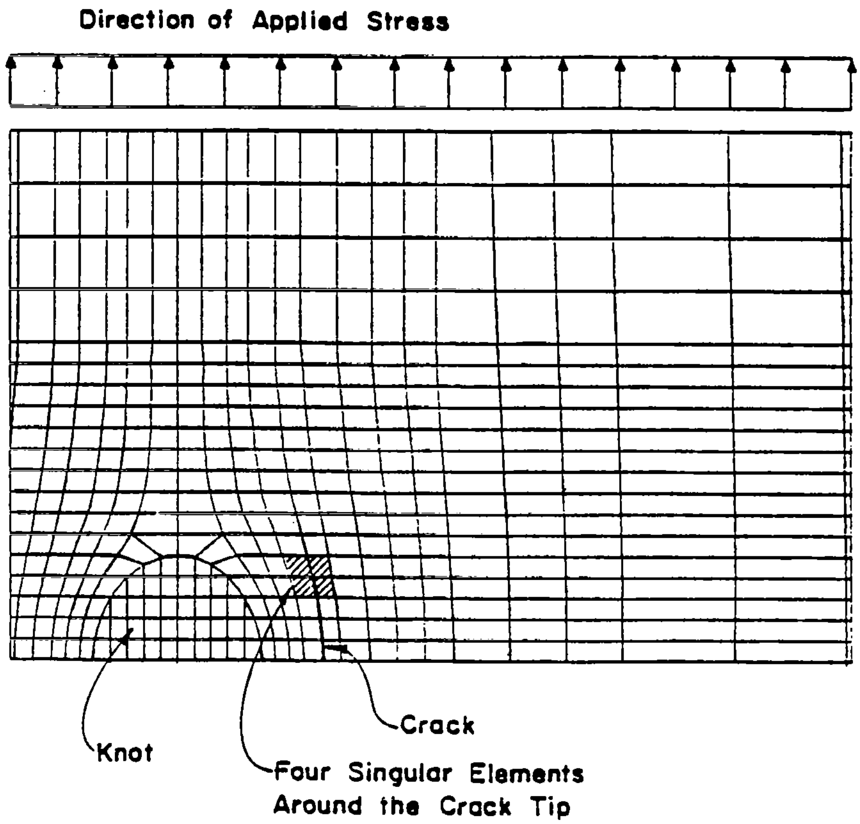

Figure 2. Example Finite Element Mesh Including Crack

The output directly calculated from each analysis is as follows:

1) Horizontal and vertical displacement at each node in the mesh.

2) Stresses for each element, parallel-to-grain, perpendicular-to-grain, and shear.

3) Stress intensity factors resulting from the use of singular elements.

4) A failure summary that indicates to the user what appropriate action should be taken to model the next step in the failure process.

The stress intensity factors directly reflect the strength of the stress field around the crack tip. The stress intensity factors are compared within the program to a fracture criteria for structural wood members to determine if the existing crack propagates at a given applied load. The element stresses are compared to a failure criteria for structural wood members to determine if a crack will form near the element under consideration. The results of these comparisons are expressed in the failure summary.

Analyses are performed repeatedly with stress and stress intensity factors monitored at each step and compared within the program logic to the fracture/failure criteria. As the load on the member is increased, more cracking and material failure occurs. The user, based on the information in the failure summary and the overall stress picture, gives the program the necessary information to model the successive step in the failure process. In the future, as research progresses, program logic will be expanded to include the decision making process the user currently makes based on the failure summary. Failure may be continually modeled in this fashion until the member under consideration has failed to the point where it cannot resist an increase in load. At this point, the predicted strength is realized. In studying the behavior of a wood member, 30 analyses may typically be performed before the member reaches its capacity. A simplified diagram of the failure model is contained in Fig. 3.

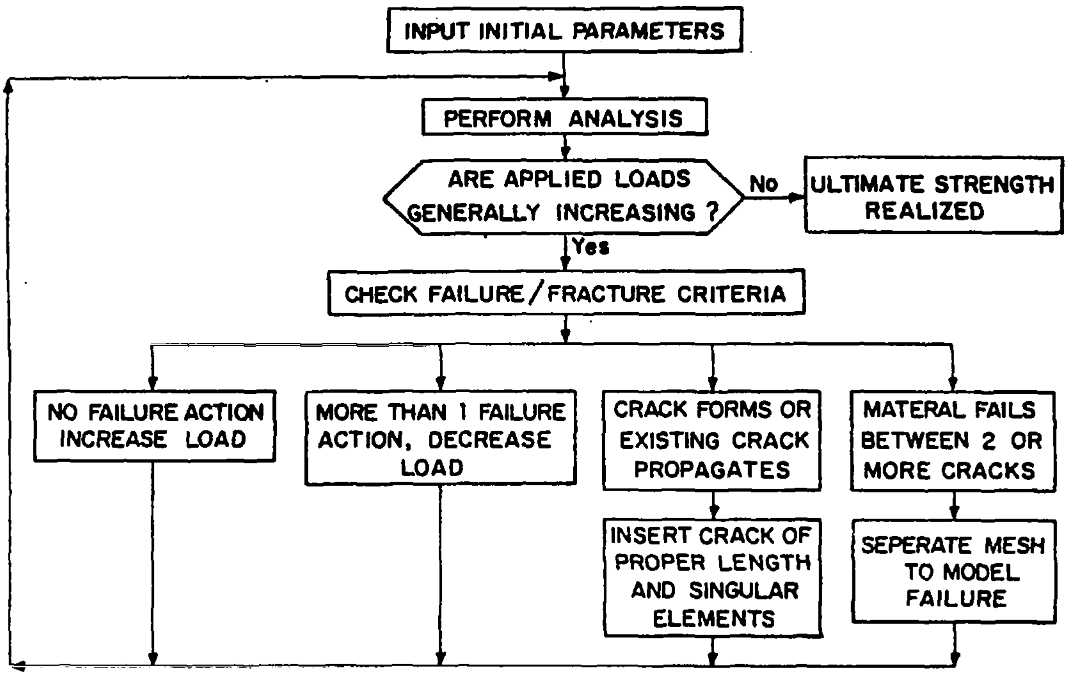

Figure 3. Strength Prediction Model

ASPECTS AND IMPLICATIONS OF VECTORIZATION

For each analysis, program STARW performs five general sets of computations:

1) Generation of a suitable finite element mesh using the flow-grain analogy and an unzipping process to include cracks.

2) Formation of a set of simultaneous equations which may be 2000 to 5000 equations in length.

3) Solution of the simultaneous equations using Gauss elimination.

4) Calculation and coordinate transformation of element stresses based on the solution vector and the element grain angles.

5) Computations with the failure/fracture criteria using element stresses and stress intensity factors as input.

Routines included in items 1 through 4 existed in limited form and were executed for small problems on a CYBER 720 prior to application on the CYBER 205. Failure calculations in item 5 and additional mesh generation capabilities were added and designed specifically for use on the CYBER 205. After compiler induced vectorization proved to be inadequate, in significantly reducing execution time, it became apparent that it was essential to explicitly vectorize selected portions of the existing routines. At the same time, it was not the primary goal of the project to expend unlimited effort to achieve the maximum in vector processing, rather the goal was to produce a powerful research tool that could be economically implemented. The bulk of the conversion (and execution time savings) were achieved with modest effort after becoming familiar with vector processing syntax.

To date, a means to vectorize the iterative solution of the fluid mechanics equations contained in the flow-grain analogy has not been established. This is not of great concern since, as in many finite element routines, mesh generation does not account for a significant portion of the total execution time. However, the unzipping of the finite element mesh to model cracks involves, in part, a uniform renumbering of nodal points. This renumbering is easily accomplished with basic vector commands since nodal coordinates are stored in vector form.

Formation of the set of simultaneous equations can typically take from 5 to 50 per cent of total execution time in a unvectorized finite element analysis. In program STARW, a 16 by 16 element stiffness matrix must be constructed for each element and properly combined with other element stiffness matrices to form the coefficient matrix (global stiffness matrix) of the simultaneous equations. Formation of the 16 by 16 matrix involves dot products of vectors of length 16. Some time savings is attained here through the use of the CYBER Q8SDOT command even though the vector length is rather small.

Solution of the simultaneous equations typically requires 40 to 90 percent of the total execution time of a finite element analysis. The 90 percent figure is not uncommon for large two-dimensional analyses. Therefore, large time savings can be attained by vectorizing the solution algorithm alone. In program STARW, Gauss elimination is used to decompose the global stiffness matrix, followed by a back substitution to obtain the solution. For the problem under consideration the stiffness matrix is banded and symmetric, and therefore, only the upper diagonal half of the matrix is stored. Furthermore, if the global stiffness matrix is stored in columns rather than rows, then adjacent terms in a row of the global stiffness matrix will be stored contiguously. Since Gauss elimination involves operations of one row upon another, by storing the matrix as described, each row will be a vector. "Gather" and "scatter" vector formation commands are unnecessary. Gauss elimination involves operations on the matrix rows in a number of nested DO loops. Vectorization of even the inner most loop results in large time savings. Back substitution involves repeated dot products of previously formed vectors. This can again be easily accomplished with the CYBER Q8SDOT command. An unvectorized and otherwise identical vectorized portion of the back substitution is shown in Fig. 4 to illustrate typical vectorization.

```
      DO  460  J = 2, JEND
      J1 = I1 + J - 1
      B(I1) = B(I1) - A(J,I1) * B(J1)
460   CONTINUE
```

```
      LE = JEND - 1
      J1 = I1 + 1
      B(I1) = B(I1) - Q8SDOT (A(2, I1; LE), B(J1; LE))
```

Figure 4. Example DO Loop and Corresponding Vector Syntax

With the solution of the equations established, element strains and stresses can be calculated in global coordinates. Since this calculation is essentially the same for every element, and care is taken to store the necessary quantities in vector form, basic vector operations accomplish this task. The solution vector is found in the global coordinate system and thus the calculated stresses are also expressed in this system. It is desireable, however, to know the stresses in the coordinate system of each element or the perpendicular-to-grain and parallel-to-grain directions. The element stresses must be transformed according to the element grain angle. Since the element grain angles are stored contiguously and in order, this computation can be accomplished with basic vector commands.

To complete an analysis, the stresses and stress intensity factors for cracks must be inserted into the failure/fracture criteria. The failure/fracture criteria interfaces the mathematical results from an analysis to the real life failure actions. Required information includes the maximum stresses and their locations within the flow-grain mesh. Since stresses are stored in element order in vectors, this information can be obtained much quicker and more easily by using CYBER Q8 commands than with scalar search algorithms.

To put the vectorization discussed into perspective, a typical problem was analyzed using unvectorized and vectorized routines. Since unvectorized versions of the mesh generator (item #1) and the maximum stress searching routine (item #5) do not exist, vectorized routines had to be used for both sides of the example. The example problem consisted of 4180 degrees of freedom (equations) and for simplification no cracks were included. The corresponding CPU execution times for different phases of the analysis are shown in Table 1.

TABLE 1. EFFICIENCY OF EXECUTION TIME FOR VECTORIZED ROUTINES

	UNVECTORIZED TIME IN SEC.	VECTORIZED TIME IN SEC.	EFFICIENCY UNVECT/VECT
MESH GENERATION	1.90	1.90	1.00
STIFFNESS MATRIX FORMATION	4.84	2.80	1.73
SOLUTION OF EQUATIONS	97.87	4.91	19.90
MISCELLANEOUS COMPUTATIONS	5.05	4.60	1.10
TOTAL	109.66	14.21	7.70

As clearly shown for this problem, the vectorized equation solver was 20 times faster than its otherwise identical unvectorized version. This savings, along with other vectorization, reduced analysis time by nearly a factor of eight. One will note that while the miscellaneous computations were somewhat insignificant in the unvectorized analysis, they take on new importance in the vectorized analysis. Additional effort may be well spent in further vectorization of the miscellaneous computations.

CONCLUSIONS

Failure in wood members is being successfully modeled and analytically investigated in greater detail than before possible through implementation of program STARW on the CYBER 205 (2). An understanding of material failure is essential to accurately predict member strength and to safely and efficiently use the material in engineering application.

Vectorization of program STARW has reduced an unwieldly and expensive, nonlinear failure modeling method into an efficient research tool. Vectorization of existing routines need not be a lengthy and laborious effort to achieve execution time savings. It has been shown that careful organization of operands into vectors and modest effort in invoking vector syntax can cut program execution time by a factor of nearly 8 for a typical problem in this research. The largest savings is realized in the solution of the simultaneous equations.

While use of program STARW is expected to provide new information on fracture and failure in wood members, the availability of machines with the capabilities of the CYBER 205, in general holds promise for advances in the analytical modeling of all materials. These advances in research will initiate new applications of materials and more efficient and reliable use of materials in existing applications.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the support of this research by the National Science Foundation under Grant No. CME-79-18170 and the Institute for Computational Studies at Colorado State University.

REFERENCES

1. Atluri, S. N., A. S. Kobayshi and M. Nakagaki, "An Assumed Displacement Hybrid Finite Element Model for Linear Fracture Mechanics", Int. Journal of Fracture Mechanics, Vol. 10, p. 1281-1287, 1975.

2. Cramer, S. M., "Analytical Strength and Fracture Prediction in Lumber", Ph.D. Dissertation in progress, Civil Engineering Department, Colorado State University, Fort Collins, Colorado, 1983.

3. Goodman, J. R. and J. Bodig, "Tension Behavior of Wood - An Anisotropic Inhomogeneous Material", Final Report to the National Science Foundation (Grant No. ENG 76-84421), Structural Research Report No. 32, Department of Civil Engineering, Colorado State University, Fort Collins, Colorado, 1979.

4. Phillips, G. E., J. Bodig and J. R. Goodman, "Flow-Grain Analogy", Wood Science, 14(2):55-64, 1981.