Proceedings of the International Conference for Agricultural Machinery & Process Engineering
October 19-22, 1993, Seoul, Korea

ADVANCED DESIGN ENVIRONMENT WITH ADAPTIVE AND KNOWLEDGE-BASED FINITE ELEMENTS

Kamyar Haghighi and Eun Kang

Department of Agricultural Engineering
Purdue University
West Lafayette, IN 47907, U.S.A.

ABSTRACT

An advanced design environment, which is based on adaptive and knowledge-based finite elements (INTELMESH), has been developed. Unlike other approaches, INTELMESH incorporates the information about the object geometry *as well as* the boundary and loading conditions to generate an *a-priori* finite element mesh which is more refined around the critical regions of the problem domain. INTELMESH is designed for planar domains and axisymmetric 3-D structures of elasticity and heat transfer subjected to mechanical and thermal loading. It intelligently identifies the critical regions/points in the problem domain and utilizes the new concepts of substructuring and wave propagation to choose the proper mesh size for them. INTELMESH generates well-shaped triangular elements by applying triangulation and Laplacian smoothing procedures. The adaptive analysis involves the initial finite element analysis and an efficient *a-posteriori* error analysis and estimation. Once a problem is defined, the system automatically builds a finite element model and analyzes the problem through an automatic iterative process until the error reaches a desired level. It has been shown that the proposed approach which initiates the process with an *a-priori*, and near optimum mesh of the object, converges to the desired accuracy in less time and at less cost. Such an advanced design/analysis environment will provide the capability for rapid product development and reducing the design cycle time and cost.

Key Word : Finite Element Mesh Generation, Knowledge-Based Expert System,
Adaptive Finite Element Analysis

INTRODUCTION

The finite element method (FEM) today is the standard procedure for most engineering analysis and design problems. Because of its versatility, FEM is the computational basis and the main component of many computer-aided design systems. However, successful use of the technique still requires significant expertise, time and cost. Many researchers are investigating ways to further automate this analysis technique, thus allowing improved productivity, more accurate solutions, and use by less trained personnel.

In FEM, often the most time consuming and expertise-intensive task faced by an analyst is the discretization of a general geometric definition of the problem into valid finite elements. With the increasing complexity of today's analysis tools, these can only be satisfied by engineers or scientists who make use of their special analytical skills.

Similarly, the usefulness of the results obtained by these methods will depend heavily on the user's fund of knowledge and experience in this area and the complexity of the finite element model being investigated. Therefore, the user controlled interactive generation of finite element models is error prone. In addition, humans are relatively slow and expensive, and their continued involvement in all phases of this process will cause it to become a worse bottleneck in the design process than it already is.

To improve this kind of user-dependent analysis/design process, some mesh generators of varying degrees of automation have been developed. But *all* of the existing automatic mesh generators *only* consider the geometric shape of the object and do not account for the boundary and loading conditions, to generate the initial mesh. This means that these mesh generators start the process with a coarse mesh which results in larger levels of error and therefore requires more adaptive steps to converge to a desired accuracy. In other words, this process is more costly and time consuming. This study presents an efficient and automatic mesh generation procedure that incorporates the information about the object geometry *as well as* the boundary and loading conditions to generate an *a-priori* (before the finite element analysis is carried out) mesh which is more refined around the critical regions (singularities, holes, re-entrant corners, ...) of the object. This would mean that the analysis process would be more efficient, yielding more accurate solutions.

Artificial intelligence has recently opened new possibilities for mechanical design. This technology holds great promise for increasing the productivity of the design community, but its full potential has yet to be realized. The research community has developed quite a few prototype design tools based on expert systems technology. In this research, artificial intelligence (AI) is successfully employed to handle the information of object geometry *as well as* the boundary and loading conditions.

Incorporation of artificial intelligence techniques, automatic meshing procedures, and adaptive analysis will greatly enhance the capabilities of the finite element technology. Such an advanced design/analysis environment will provide the capability for rapid product development and reducing the design cycle time while simultaneously increasing the number of design alternatives which might be overlooked in a conventional design approach.

METHODOLOGY

In this work, a blackboard architecture expert system (Corby, 1986) is developed to intelligently identify critical regions/points which need further refinement and to choose the proper mesh size for them. This architecture treats problem-solving as an incremental, opportunistic process of assembling a satisfactory and optimal configuration of solutions developed by different knowledge sources.

In this section, procedures for developing an automatic *a-priori* (before the finite element analysis is carried out), and near optimum initial mesh of the object in two-dimensional linear elasticity problems are described. This is in contrast to the existing automatic mesh generators that start the process with an initial coarse mesh. The philosophy behind the proposed approach is based on the premises in the paper by Kang and Haghighi (1992).

Graphical Interface

The graphical user interface is menu driven and utilizes multiple command and display windows to create and edit the object description interactively. For the purpose of automatic mesh generation, the user can specify the geometric definition of the object by simply using the mouse or a data file. The object shape data, its features (line segments, arcs, circles), and their geometric characteristics (location, slope, center point, radius, ...) are stored in a counter clockwise fashion for the exterior boundary and in a clockwise fashion for the interior boundary. Next, boundary attributes such as boundary support data (fixed, hinge, and roller supports, and their locations) and loading data (distributed load, concentrated force, and pressure, with their magnitudes and locations) are stored. This information can be easily retrieved for the automatic mesh generation.

Selection and Priority of Critical Regions

The blackboard expert system first identifies candidate regions and/or points that are critical. This includes regions with geometric irregularities, points or regions of boundary support and loading, and regions where material behavior and properties change. This process is accomplished by a "Critical Region " Knowledge Source (KS) that detects regions and/or points in which a significant stress gradient might be developed such as around notches, holes, and cracks.

Once all the candidates for critical points have been identified, the blackboard system performs a useful force analysis to select only those candidate points which are truly critical, after incorporating the effect of various forces and points of boundary support. This is done by a "Boundary Condition" KS that handles various kinds of loading and boundary conditions and their contribution to identify the "true" critical points.

The next step involves decomposition of the original structure (or domain) into substructures (or standard cases) for which an initial and approximate stress concentration calculations can be performed. This calculation is based on either using the available and standard cases of which analytical solutions exist or using heuristics. This is done by a "Standard Case" KS.

The usefulness of this approach should not be measured by how accurately the solutions around the critical points are predicted initially, but should be measured by the ability of the procedure to allow the automatic development of finite element meshes yielding results to the desired level of accuracy in less time and at less cost. It is anticipated that the above procedure could prove quite useful within the framework of an adaptive analysis process.

By comparing the estimated stress values of critical regions(points/surfaces), "priorities" are then assigned to them. The critical region with the maximum stress value is labeled "most critical" and receives highest "priority." This means that the area around this critical region has the smallest mesh size. The largest mesh size is assigned to regions that have uniform nominal stresses. The next largest mesh size will go to the critical region with the least priority value. Through this process, a priority value and its corresponding mesh size will be assigned to each critical region by the blackboard system.

Generation of Elements

The determination of mesh sizes around each critical region is accomplished through a "Wave Propagation" Knowledge Source (KS) using the "wave propagation" concept (Kang and Haghighi, 1992). This concept describes refinement waves that are generated from each critical point serving as the center of waves. The "Wave Propagation" KS provides the "wave spacing" and "wave priority" that are related to the degree of mesh refinement. The wave propagation concept is fully automatic and does not require any user-provided information except the data which defines the object geometry and boundary conditions. Therefore, the possibility of generating ill-shaped elements and rapid change of mesh size is eliminated.

Once the wave spacing and priority for all the critical points are established and waves are generated, the position of the nodes on the waves and their coordinates are obtained according to the procedure proposed in the paper by Kang and Haghighi (1992). Three different cases that are treated include node generation around critical points, node generation around critical surfaces and node generation on the boundaries.

Once nodes have been generated for the entire domain, boundary segments are defined by sequentially connecting the nodes which are located on the boundary. The boundary segments on the exterior boundary are connected in a counter clockwise fashion while the boundary segments on the interior boundary are connected in a clockwise fashion. These boundary segments are used as base segments for generating triangular elements. In this work, a new and efficient algorithm for element generation (Kang and Haghighi, 1992) has been used that needs to check only the nodes and generation front boundary segments which are in a certain range, instead of checking all the nodes and generation front segments in the entire domain, as was proposed by Lo (1985). A search is performed to locate a third node which forms the element closest to an equilateral triangle with the same base segment. For this purpose, the properties of the Delaunay triangulation (Lo, 1985; Sibson, 1978) is used.

When the entire domain is completely triangulated, Laplacian smoothing (Lo, 1985; Yerry and Shephard, 1983) procedure is applied to improve the mesh. This is achieved by perturbing the triangulation so that elements are more closely equilateral triangles. Each interior node of the triangulation is replaced by the centroid of the polygon composed of those triangles which surround that node. This improves the elements aspect ratios. The smoothing process is applied several times to the entire mesh of the structure until no significant change is detected in the nodal coordinates.

Adaptive Mesh Improvement

After the initial near-optimal mesh based on *a-priori* information has been generated, the finite element system of equations will be constructed and solved. During the calculations which are aimed at the assessment of the quality of meshes, error indicators are computed for each element together with an estimate of the global error (Zienkiewicz and Zhu, 1991). If the global error exceeds a specified limit, the system calls for refinement and reanalysis. This process continues automatically until the global error estimates fall below a specified limit. The flowchart of the process is shown in Figure 1.

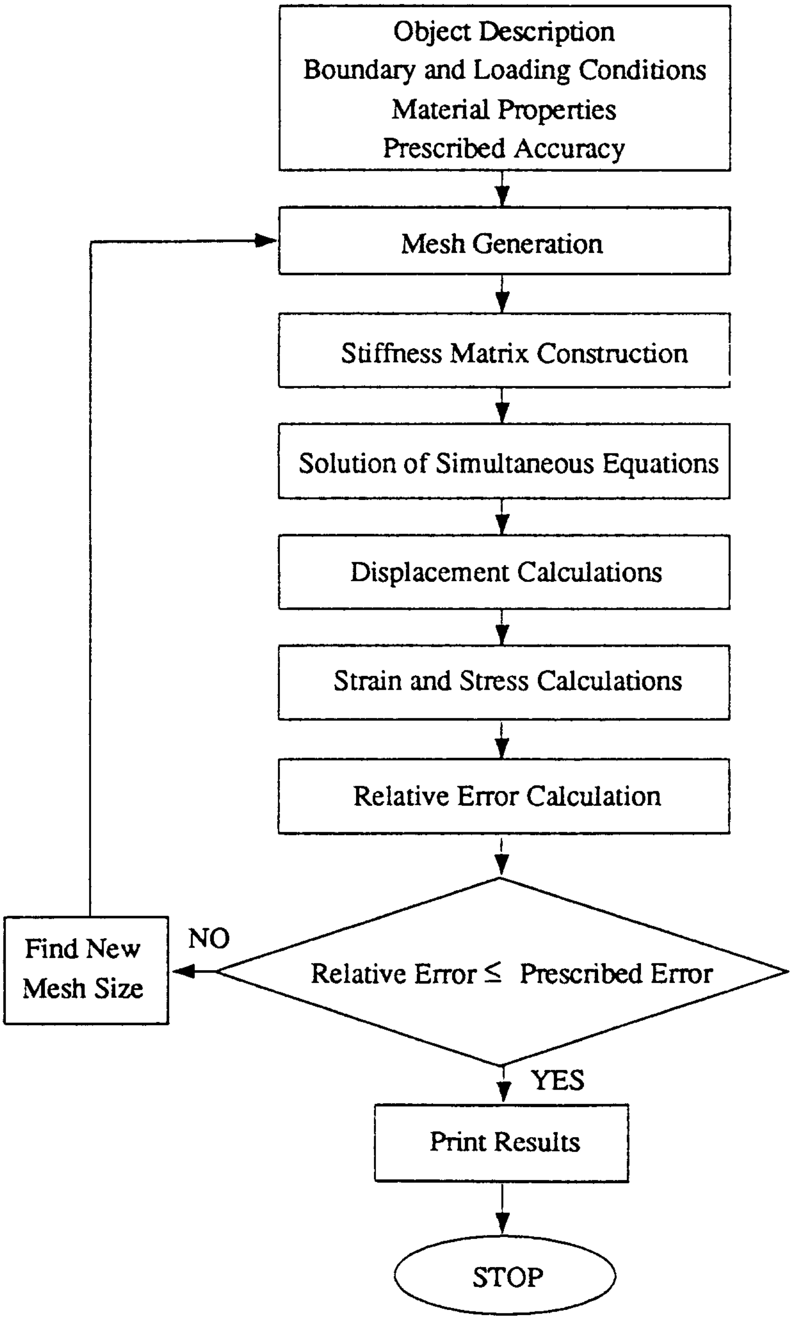

Figure 1. Flowchart of the adaptive refinement process

IMPLEMENTATION

To illustrate the performance of the proposed environment (INTELMESH) in terms of the quality of the initial mesh and its efficiency for adaptive process, a rectangular plate with a center hole is analyzed. The geometrical dimensions and the boundary conditions are shown in Figure 2. The problem is that of plane stress conditions. Because of symmetry of both the plate and the boundary conditions only a quarter of the plate, shown darker in Figure 2, needs to be modeled. The prescribed accuracy is 5% relative error. For the conventional approach, an initial uniform mesh (Figure 3) is constructed to sufficiently model the geometry resulting in a relative error value of η = 31.7%. The desired accuracy is achieved after two more adaptive refinement processes (Figure 4), resulting in η = 4.33%. For the proposed approach, the finite element approximation starts on a *a-priori* refined mesh (Figure 5) with relative error η = 10.6%. For this approach, only one adaptive refinement is needed resulting in η = 4.78% (Figure 6). The proposed approach has been tested successfully for several examples and their implementation has resulted in faster convergence to the desired accuracy.

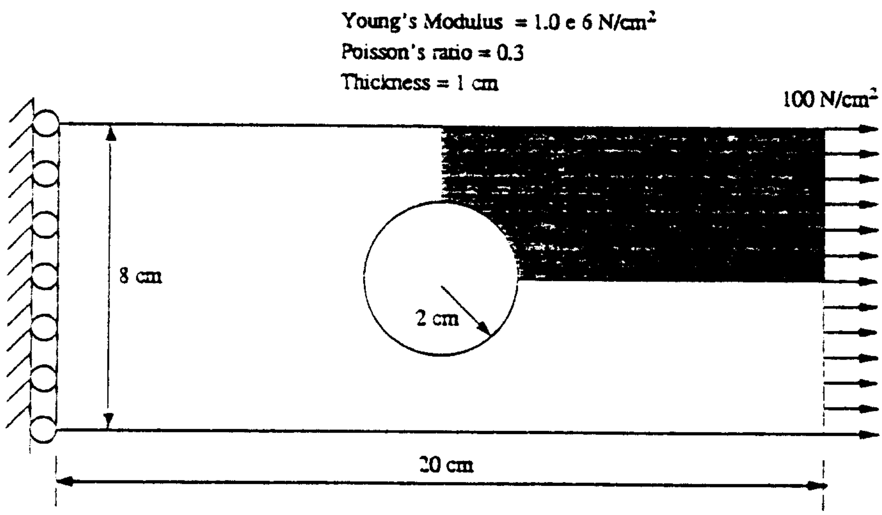

Figure 2. The schematic diagram of the rectangular plate with a circular hole

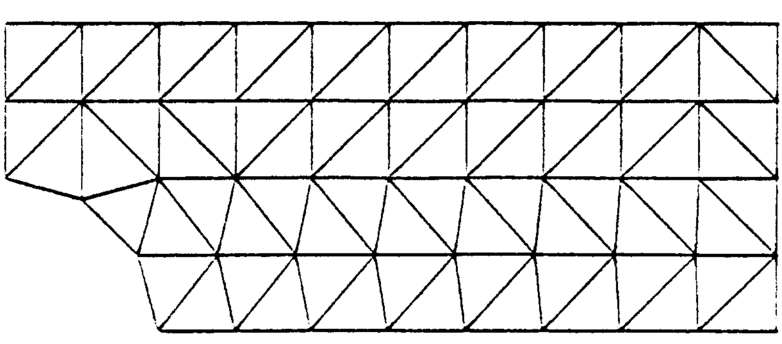

Figure 3. Uniform initial mesh, relative error = 31.7% - Conventional approach

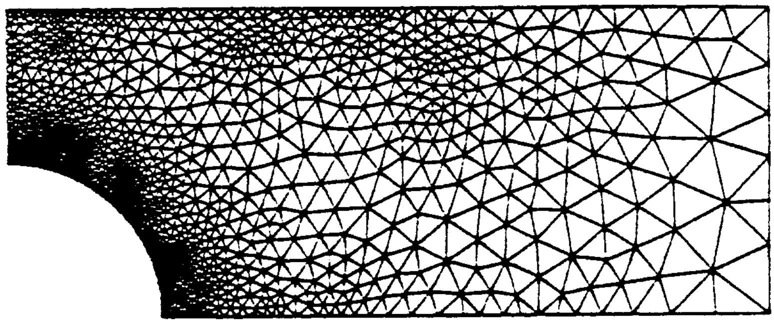

Figure 4. Adaptive mesh after 2nd refinement process, relative error = 4.33% - Conventional approach

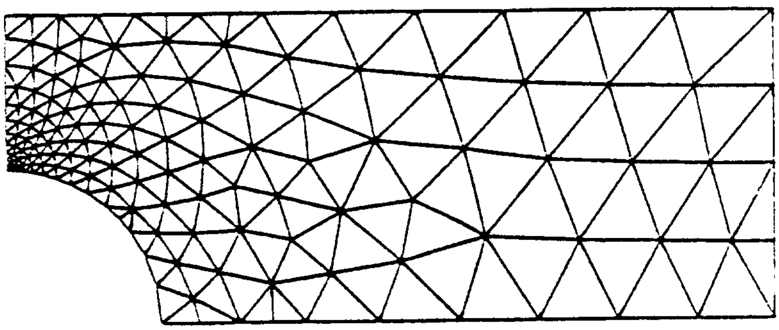

Figure 5. *A-priori* refined initial mesh, relative error = 10.6% - Proposed approach

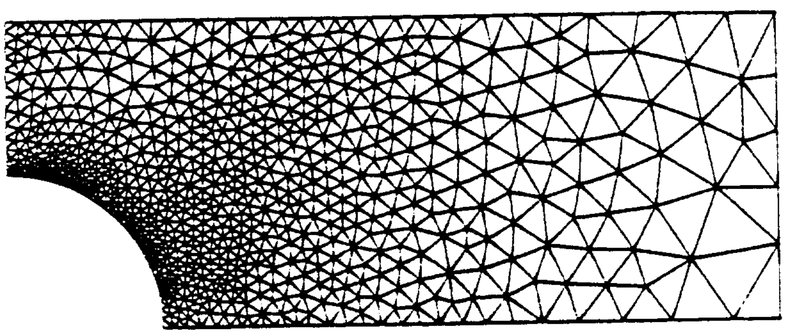

Figure 6. Adaptive mesh after 1st refinement process, relative error = 4.78% - Proposed approach

CONCLUSIONS

An advanced design environment with adaptive and knowledge-based finite elements (INTELMESH) was developed on Sun 4 Sparcstation. INTELMESH is capable of identifying the critical surfaces/points in the domain through a substructuring process. Based on the initial and approximate solution, critical points/surfaces are ranked and their mesh sizes are assigned. INTELMESH then generates an *a-priori* near-optimal graded mesh which is more refined around the critical regions of the problem domain.

The adaptive process has been successfully incorporated into INTELMESH. The use of the adaptive process in conjunction with INTELMESH provides a fully automatic finite element analysis process that can identify and reduce the errors without any user intervention. It has been shown that the proposed approach converges to the desired accuracy in less time. In addition, the use of the system does not require any special expertise and is very well suited for use by less trained finite element analysis.

REFERENCES

1. Corby, O. 1986. Blackboard Architecture in Computer Aided Engineering, Int. J. A.I. in Eng., 1(2), 95-98.
2. Kang, E. and K. Haghighi. 1992. A Knowledge-Based *A-Priori* Approach to Mesh Generation in Thermal Problems, Int. J. Num. Meth. Eng., 35:915-937.
3. Lo, S. H. 1985. A new mesh generation scheme for arbitrary planar domains. Int. J. Num. Meth. Eng., 21:1403-1426.
4. Sibson, R. 1978. Locally Equiangular Triangulations, Comp. J. 21, 243-245.
5. Yerry, M. A. and M. S. Shephard. 1983. A modified quadtree approach to finite element mesh generation. IEEE J., Jan/Feb., pp.39-46.
6. Zienkiewicz, O. C. and J. Z. Zhu. 1991. Adaptivity and Mesh Generation. Int. J. Num. Meth. Eng., 32:783-810