Nuclear Engineering and Design 98 (1986) 57–67
North-Holland, Amsterdam

FINITE ELEMENTS IN CAD AND ADINA *

Klaus-Jürgen BATHE **

Massachusetts Institute of Technology, Cambridge, MA 02139, USA

Received 22 May 1985

The use of finite element methods in computer-aided-design – CAD – is discussed. Some current capabilities are presented and important future developments are outlined. The discussion focusses on the use of the ADINA program in CAD applications.

1. Introduction

The use of finite element methods in mechanical and structural designs is now well established [1]. Finite element computer programs have been employed extensively in various engineering practices for about two decades, but during recent years, the integrated approach of computer-aided-design (CAD) has given a specific impetus to finite element analysis – and CAD is bound to influence the application of finite element methods even more strongly [2,3]. In general, CAD capabilities have increased a great deal during the recent years because of the dramatic surge in hardware capabilities available at reasonable costs, the development of very powerful software and the documented benefits reached in industry when CAD procedures are introduced.

Fig. 1 summarizes schematically the field of CAD/CAM: computer-aided-design and computer-aided-manufacturing. As illustrated, this field is a synthesis of various large areas, including finite element analysis. Fig. 1 implies, at least to a high degree, a fully automatic process of design and manufacturing. While in some engineering environments much has been achieved towards this aim, in general, the interfacing between the various fields is difficult due to the assumptions necessary in each stage of the process and the required transfer of large amounts of data and information. However, the integrated approach illustrated in fig. 1 can be the overall aim in many engineering environments and should be possible to achieve in the not too distant future.

In this paper we discuss the step of finite element analysis. This step is in more detail summarized in fig. 2: a geometric modeler is used to create the geometry of the component to be analyzed, a finite element pre-processor is employed to generate the finite element mesh and data, the solution of the finite element model is obtained, and finally the calculated solution data are evaluated using a post-processor. Fig. 2 lists some well-known example software that can be employed for the complete analysis process. Note that for the pre- and post-processing it can be effective to employ a combination of available software. This yields flexibility to the user with respect to the most efficient way to generate the input data and to display the calculated results.

When performing a finite element analysis, in essence, the process of fig. 3 is followed. Referring to this figure, the physical problem considered is the structural component subjected to certain loads. The mechanical idealization involves static and kinematic assumptions for analysis purposes that together lead to differential equations governing the structural model. The finite element analysis then solves this model. This analysis requires an assessment of accuracy; if the accuracy criteria are not met, the finite element mesh is refined and the solution parameters are changed until an accurate solution of the mechanical model is obtained. At this point the results can be interpreted in a meaningful manner for the design of the component. As summarized in fig. 3, it may next be necessary to consider a more refined mechanical model, or design improve-

* Invited Division Lecture at the 8th International Conference on Structural Mechanics in Reactor Technology, Brussels, Aug. 1985.

** Professor of Mechanical Engineering.

0029-5493/86/$03.50 © Elsevier Science Publishers B.V.
(North-Holland Physics Publishing Division)

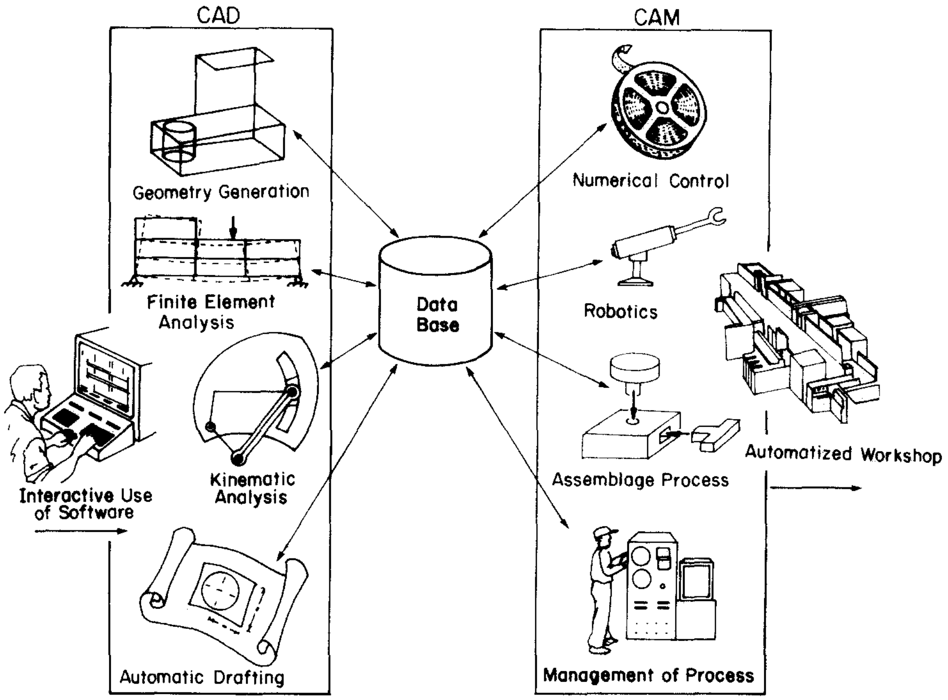


Fig. 1. The field of CAD/CAM viewed schematically [4].

- Geometric modeler is used to generate geometry

  MEDUSA, EUCLID, GEOMOD,...

  ↓ geometry

- Finite element pre-processors are used to generate finite element mesh and data

  PATRAN, SUPERTAB, FEMGEN,...

  ANSYS, NASTRAN, ADINA-IN,...

  ↓ finite element data

- Finite element solution is performed

  ANSYS, NASTRAN, ADINA,...

  ↓

- Finite element post-processors are used to evaluate solution results

  PATRAN, SUPERTAB, MOVIE.BYU,...

  ANSYS, NASTRAN, ADINA-PLOT,...

Fig. 2. Geometry generation and finite element analysis in CAD.

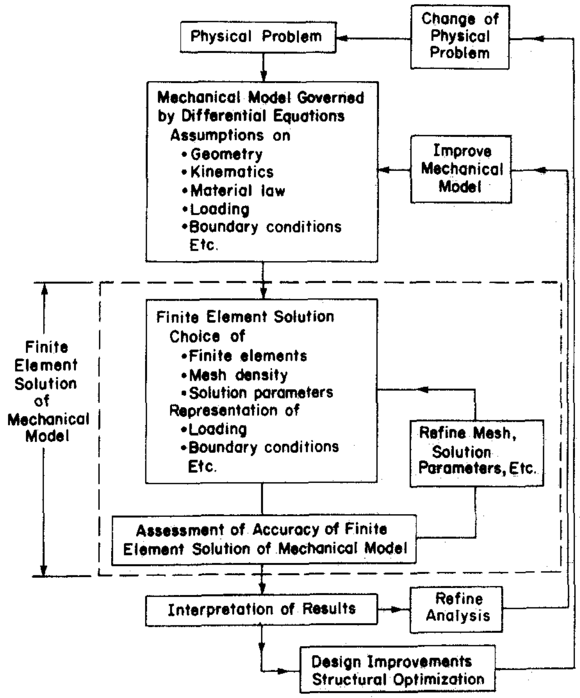


Fig. 3. The process of finite element analysis.

ments in the search for a structural optimization which all leads to further finite element solutions.

The finite element system used in this process surely should have the desirable characteristics summarized in fig. 4.

In conventional finite element analysis, the finite element solution of the mechanical model (see dashed box in fig. 3) is largely performed by the analyst based on experiences with the numerical procedures used. This may require that the analyst be quite familiar with the finite element techniques, and such needed experience can place an undue burden on the designer or analyst. The major impact of finite element analysis in the CAD environment will occur when the finite element analysis of a mechanical model – i.e., the choice of finite elements, mesh density, solution parameters, the assessment of solution accuracy and the refinement of the mesh until a satisfactory accuracy level has been reached – is to a high degree performed automatically by the finite element software [5]. This capability can, in principle, be provided for many types of applications; namely, for many linear static and linear dynamic analyses.

However, although the aim must be to automatize the finite element analysis as much as possible so that the designer is free to "play with ideas" in the design process, it should be emphasized that these automatic procedures cannot be expected to replace the creativity of the engineer. Instead, the procedures will provide increasingly more powerful tools that allow the designer to investigate, in an ever more flexible and creative manner, design ideas through the analysis of appropriate linear elastic mechanical models. If a nonlinear analysis is to be performed – the need for such must be decided by the designer, just as the mechanical model must be selected – then, depending on the complexity, the designer/analyst must also draw on considerable knowledge and creativity to establish effective analysis models.

- Ease of problem definition
- Ease of geometry generation
- Ease of meshing
- Linear, geometric and material nonlinear solution
- Ease of evaluating results
- Optimization

Fig. 4. Desirable characteristics of a finite element system.

Our objective in this paper is to discuss the use of finite element procedures in CAD considering the current state-of-the-art and important future developments. To give focus to the discussion, the analysis capabilities of the ADINA program are briefly presented and we concentrate on the research and developments that we pursue to render ADINA an increasingly more powerful finite element tool in CAD. In the next section of the paper we briefly present the current capabilities of ADINA for CAD applications, and in the section to follow we then discuss some research and development tasks currently pursued. We conclude the paper with some summarizing remarks on the future of finite element procedures in CAD.

2. Finite element analysis with ADINA

The name ADINA stands for *A*utomatic *D*ynamic *I*ncremental *N*onlinear *A*nalysis. This name reflects the aim to have a comprehensive finite element program that will perform the finite element analysis of a mechanical model in an ever increasing *automatic* manner, as needed in CAD applications.

The complete ADINA system consists of the program ADINA for displacement and stress analysis, ADINAT for analysis of heat transfer and field problems, ADINAF for analysis of low Reynolds number flow with heat transfer and the pre- and post-processors ADINA-IN and ADINA-PLOT [6,7].

2.1. Finite elements and material models

The program ADINA contains a few most effective elements which can be employed to model a large variety of problems. This approach is deemed to be most appealing in practical applications. The elements are depicted in fig. 5. They can be employed for linear elastic analysis and materially and geometrically nonlinear analyses. Particularly noteworthy are for practical applications the shell analysis capabilities of the program, using the shell elements of fig. 5f.

The material models available in ADINA are summarized in fig. 6. Here the concrete model, the thermo-elasto-plastic and creep analysis capabilities, and the large strain elastic and inelastic analysis capabilities are important features.

2.2. Solution capabilities

The finite elements and material models given in figs. 5 and 6 can be employed with various general analysis options.

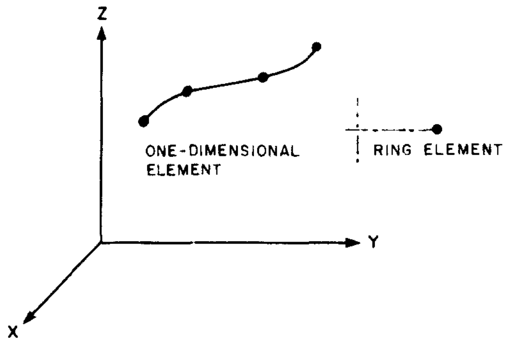


Fig. 5. Finite elements in ADINA; (a) Truss and cable element (2,3, or 4 nodes).

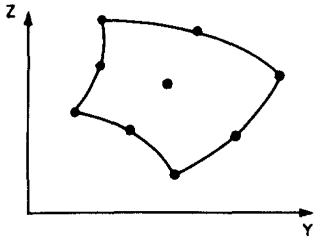


Fig. 5b. Two-dimensional solid and fluid elements (variable number of nodes).

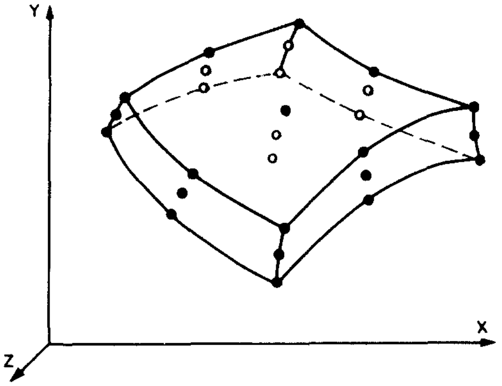


Fig. 5c. Three-dimensional solid and fluid elements (variable number of nodes).

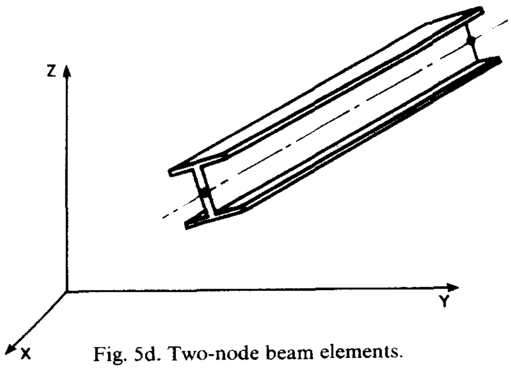


Fig. 5d. Two-node beam elements.

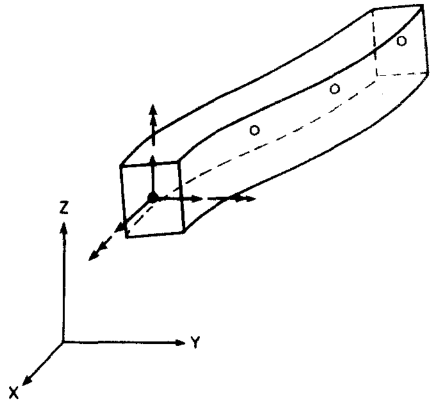


Fig. 5e. Isoparametric beam element (2, 3, 4 nodes).

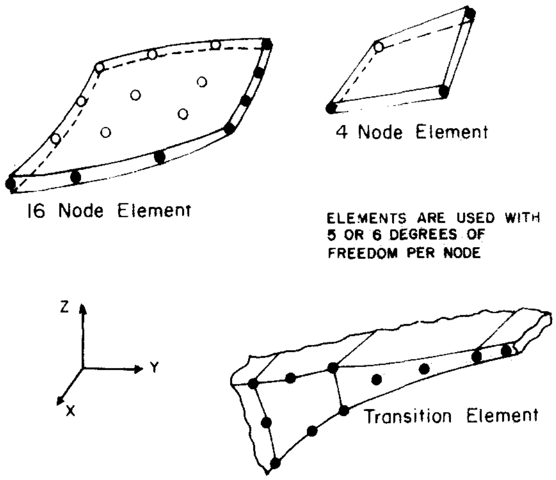


Fig. 5f. Plate and shell elements.

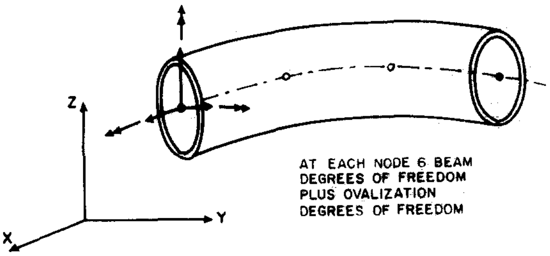


Fig. 5g. Pipe element with ovalization.

Isotropic Linear Elastic

Orthotropic Linear Elastic

Isotropic Thermo-Elastic

Curve Description Type Models for Analysis of Geological Materials

Concrete Models

Isothermal Plasticity Model

Thermo-Elastic-Plastic and Creep Models

Nonlinear Elastic, Incompressible Models

User-Supplied Models

Fig. 6. The ADINA material models.

Linear and nonlinear analysis. Statics and dynamics. A linear analysis is performed when the deformations of the model are assumed to be infinitesimally small and constant material moduli are specified. A nonlinear analysis may involve material nonlinearities and/or geometric nonlinearities due to large deformations.

In a dynamic analysis, inertia and possibly damping effects are included.

Element formulations. The elements shown in fig. 5 are displacement-based finite elements or are formulated using a mixed interpolation of displacements, strains, pressures. The mixed interpolation is employed in the formulation of the elements for analysis of plate and shell structures and incompressible media [1,8,9].

Time integration. An implicit time integration method, usually the trapezoidal rule, would generally be employed for analysis of structural vibration problems, i.e., when the system is primarily excited in a few vibration modes. The central difference explicit time integration technique is primarily used to predict wave propagation phenomena.

Solution of equations. In static analysis, implicit time integration, frequency calculations (say, for mode superposition analysis), the total stiffness matrix and (consistent) mass matrix are assembled in blocks depending on the high-speed storage available in the computer, and only the elements below the skyline are stored. A skyline out-of-core column solver is employed to obtain the solution of the equations. Of much concern is the stability and accuracy of the incremental solution of the nonlinear equations. In ADINA the solution can be obtained using the modified or full Newton-Raphson methods with or without line searches, the BFGS method, or an automatic load-stepping scheme for collapse and post-collapse response calculations [10].

Substructuring. To take advantage of some repetitive geometric and material conditions in the system under consideration, substructures of linear elements can be defined. Each substructure can be employed as a "superelement" a number of times in the complete element assemblage. This option can be effective in some linear analyses to solve large systems. However, the option has also been incorporated into ADINA for nonlinear analyses in which only local nonlinearities are encountered. In these analyses it can be efficient to statically condense out the major part of the linear degrees of freedom prior to the incremental solution of the nonlinear equations. The substructuring capability can be employed in static and dynamic lumped mass analysis (without any constraints on the lumped mass matrix to be used).

Constraint equations. In some analyses it is necessary to prescribe displacements at some nodal points, and/or impose constraints between some nodal displacement components. In ADINA nodal point displacements can be prescribed as a function of the load (time) step or can be expressed in terms of other nodal point displacements. These latter constraints are frequently simply specified as rigid links, in which case the program establishes the constraint equations automatically.

Solution of frequencies and mode shapes. Desired frequencies and corresponding vibration mode shapes can be calculated using the determinant search and an improved subspace iteration algorithm.

Mode superposition. The mode superposition method can be employed to calculate the time history response of linear systems, or to perform a response spectrum analysis. The mode superposition technique can also be used effectively for nonlinear analysis if the system contains only local nonlinearities.

Load definition. Prescribed concentrated, pressure and temperature loading can be defined; mass proportional loading and centrifugal loading can be applied, and nodal displacements can be imposed. The pressure loading can be deformation dependent. Also, an interface is provided through which the user can program an own load definition using the current displacements, velocities and so on (e.g., to define hydrodynamic loading).

Linearized buckling analysis. A linearized buckling analysis by an eigensolution can be performed at any load level. The calculated buckling mode shapes can be used to define geometric imperfections on the structural model in order to simulate nonperfect conditions in the collapse analysis.

Fracture mechanics. Stress intensity factors can be requested directly using ADINA. The results can be obtained in plane stress, plane strain, axisymmetric analyses and in three-dimensional analysis (using the 3-D solid elements).

Solution of contact problems. A very useful Lagrange multiplier/segment algorithm is available in ADINA to solve contact problems, in which the contact area is initially unknown and varies during the response. The bodies in contact may be flexible or rigid with Coulomb frictional conditions, and the bodies may undergo large deformations in sliding.

Fluid-structure interaction analysis. The response of fluid–structure systems can be solved using the 2-D and 3-D (acoustic) fluid elements referred to in fig. 5. These elements are formulated using the velocity potential ϕ with one degree of freedom per node. The total coefficient matrix including the solid-fluid coupling effects is a symmetric matrix. Bounded and infinite domains (with infinite elements, not shown in fig. 5) can be analyzed.

2.3. Use of ADINAT and ADINAF programs

The ADINAT program is typically employed to calculate the temperature and heat flow distributions in solids and structures. The calculated temperatures may then be used in ADINA to evaluate the corresponding thermal stresses. The ADINAT program can also be employed for seepage analysis and the solution of other field problems. The ADINAF program is a new development for the analysis of velocities, and pressure and temperature distributions in viscous low Reynolds number flows.

3. Current ADINA developments for CAD applications

The ADINA programs provide already a powerful tool in finite element analysis, but we are working on specific improvements to strengthen the automatic analysis features for CAD applications.

Fig. 7 gives an overview of an effective structural CAD system – the way we envisage the ADINA system to evolve. We note that the finite element solution pertains to the information listed in the box of the dashed boundary. The complete analysis consists of the fully automatic mesh generation, the input preparation using artificial intelligence-based procedures, the solution pertaining to the given data and the mesh improvements. Note that the information in the dashed box of fig. 7 corresponds to the finite element solution of the mechanical model (see dashed box in fig. 3) and that the

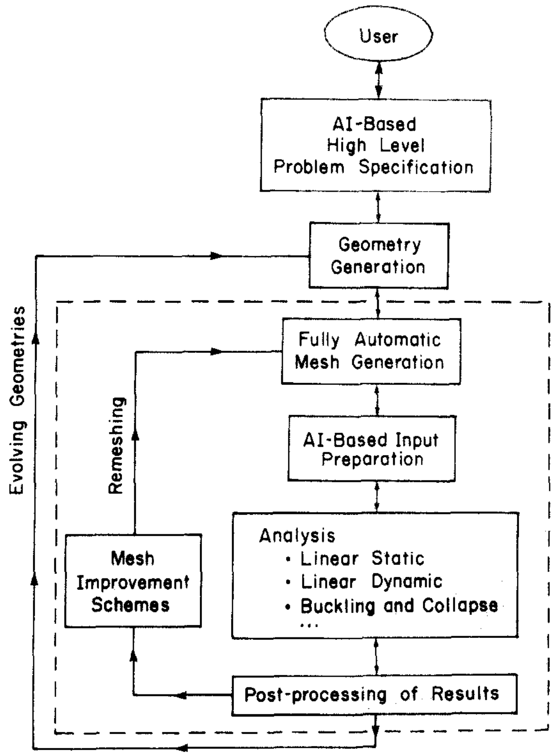


Fig. 7. Schematic of an effective finite element system in CAD.

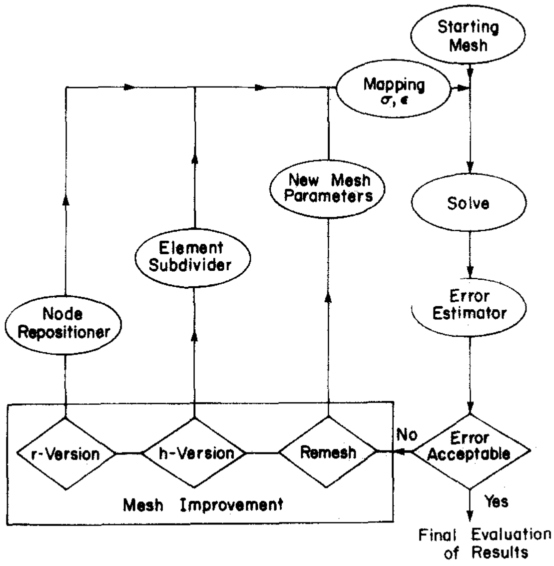


Fig. 8. Self-adaptive mesh improvement in finite element analysis.

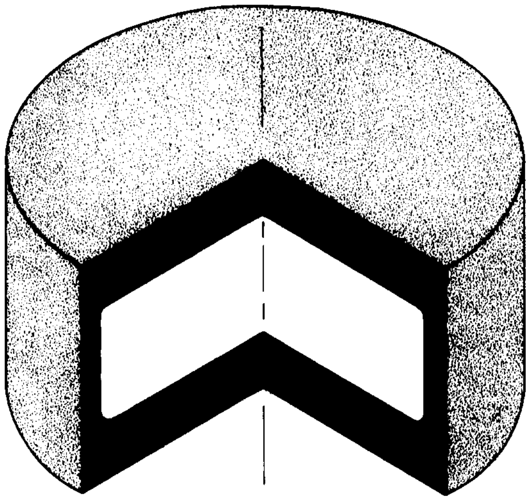


Fig. 9. Analysis of pressure vessel; (a) View of vessel.

problem specification and geometry generation in fig. 7 correspond to the creation of the mechanical model.

Considering the research and developments towards the structural CAD system summarized in fig. 7, much emphasis need still be directed to the development of ever increasingly more powerful algorithms in linear and nonlinear analysis: the solution of complex shell problems, inelastic conditions, coupled problems, rubber components, contact conditions, composite structures, nonlinear dynamic conditions and so on. These are the "traditional" areas of research and development. However, fig. 7 shows that for CAD applications most important are also the program features that help the designer to establish finite element models automatically.

The artificial intelligence-based input preparation in fig. 7 can evolve from software help in selecting certain analysis parameters (for example, the specification of boundary conditions, time or load step magnitudes,

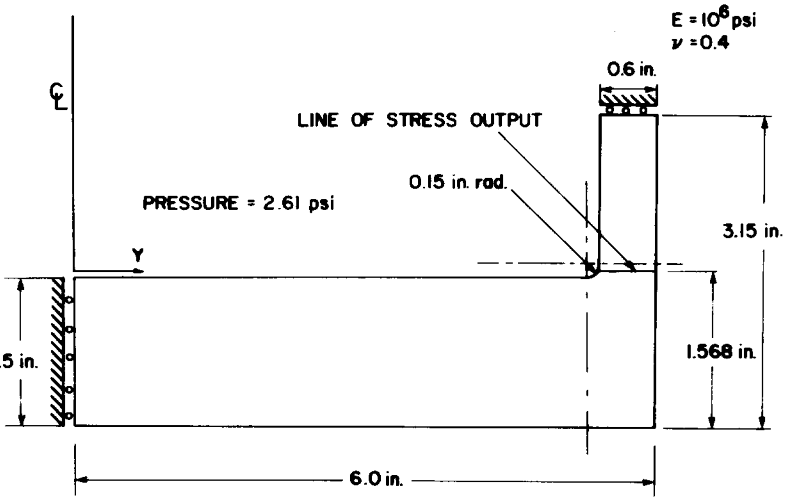


Fig. 9b. Axisymmetric model solved.

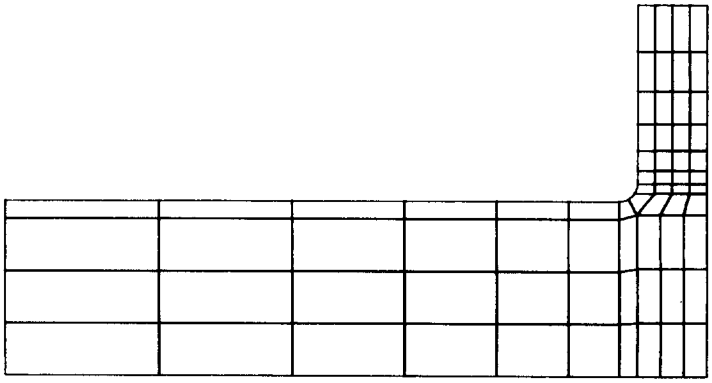


Fig. 9c. Starting mesh for automatic refinement – 69 8-node elements.

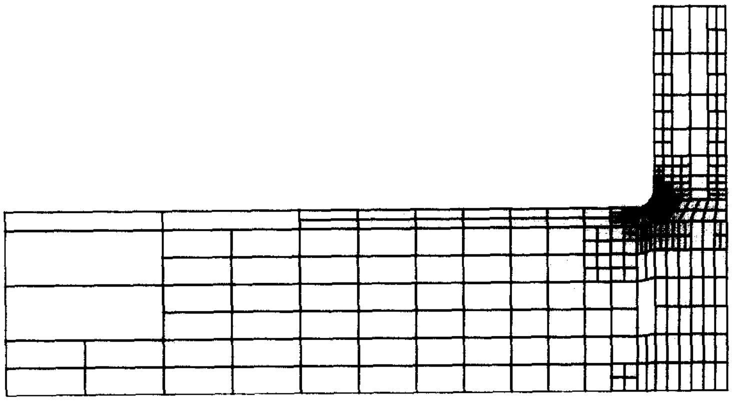


Fig. 9d. Final mesh reached in automatic refinement – 342 8-node elements.

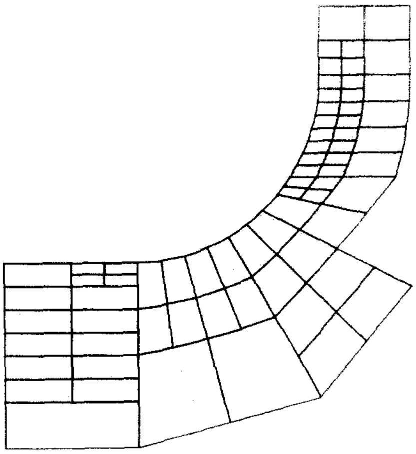


Fig. 9e. Detail of 342 element mesh.

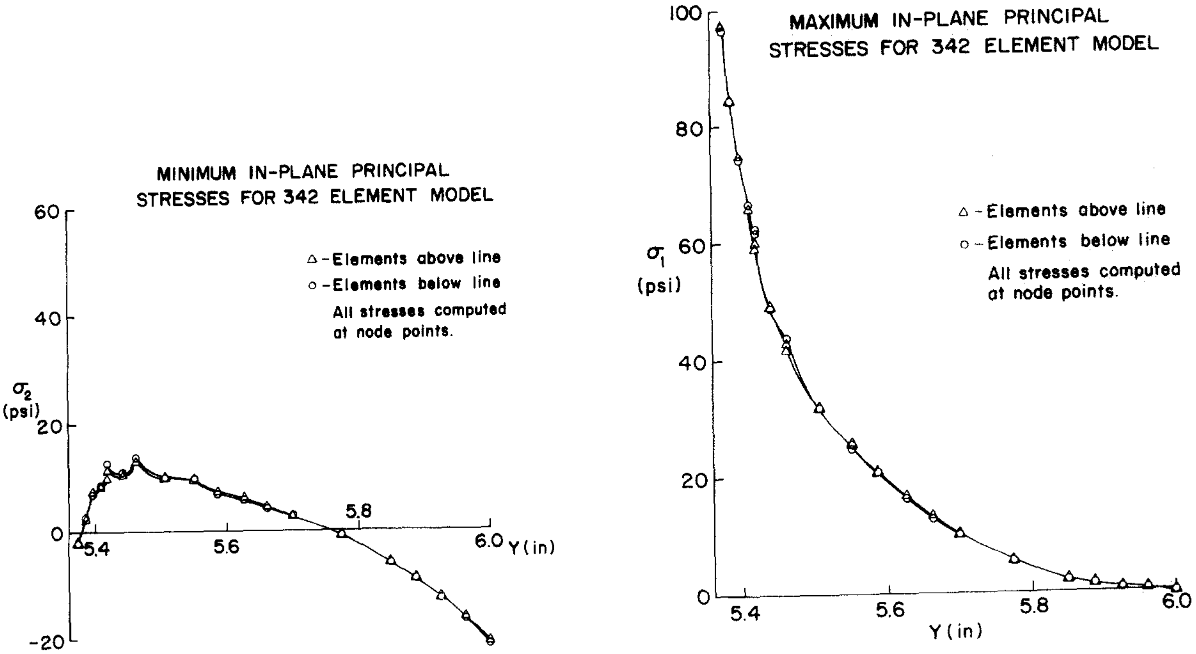


Fig. 9f. Stress predictions along line of interest shown in fig. 9b.

solution tolerances) to sophisticated help in identifying possible modeling errors. The mesh improvement schemes referred to in fig. 7 are detailed more closely in fig. 8.

Fig. 8 shows that the mesh improvement can be achieved using complete remeshing, element subdivision or node repositioning. We may note that the introduction of higher-order elements and a new mesh topology is contained in the remeshing option, and hence no specific mention of the p-version of mesh refinement is made [5].

Referring to fig. 8, we note that given the starting mesh, first the finite element solution is obtained and an error estimation is performed. If the error is un-

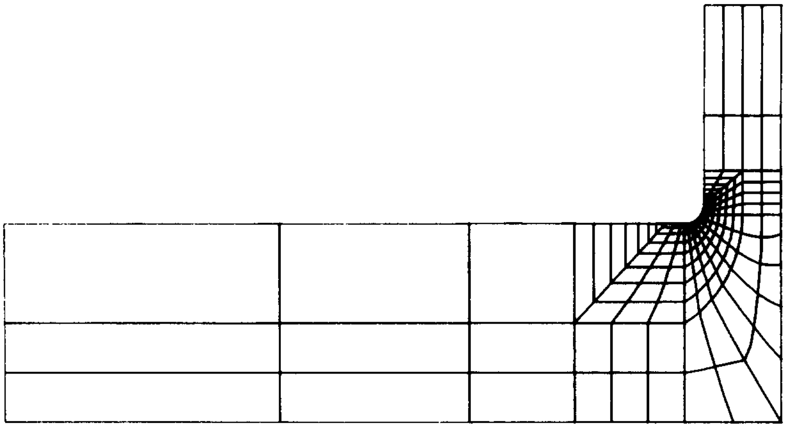


Fig. 10. Mesh of 181 8-node elements and solution results; (a) Complete mesh.

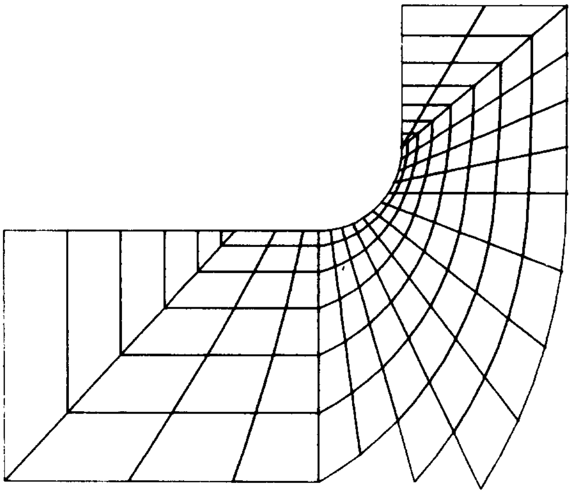
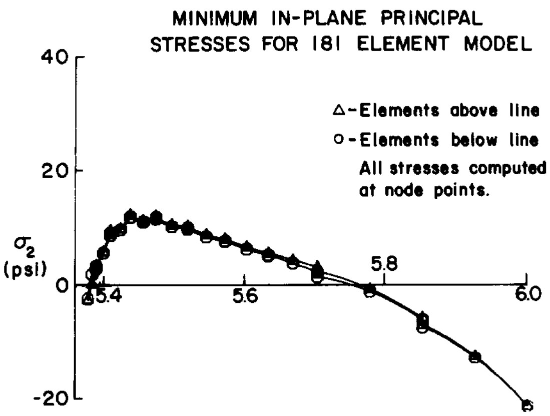


Fig. 10b. Detail of mesh.

acceptably large, the mesh improvement schemes are used. Here it is probably most effective – when the starting mesh is rather coarse – to establish by remeshing a completely new mesh that results into reasonably small errors. These errors are then further reduced by element subdivision or node repositioning.



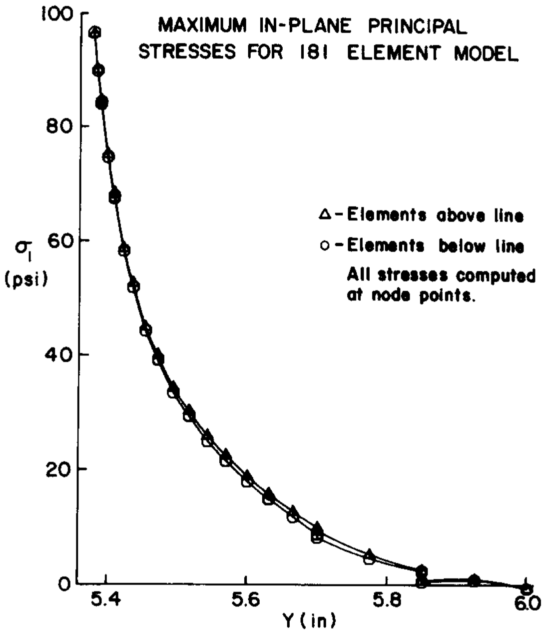


Fig. 10c. Stress predictions along line of interest, shown in fig. 9b.

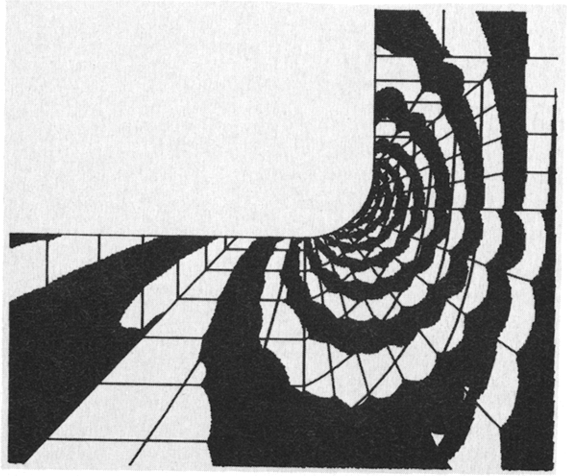


Fig. 10d. Pressure band plot near stress concentration; bands correspond to 2.5 psi.

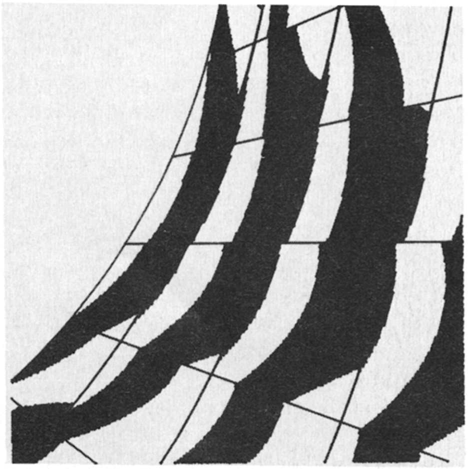


Fig. 10e. Pressure band plot at stress concentration; bands correspond to 2.5 psi.

The complete algorithm needs criteria that measure whether a further mesh refinement is necessary, which scheme is to be used and how in detail the refined mesh is to be established. The objective is to arrive at a final mesh which is optimal in terms of total computing costs expended for the required accuracy. In these decision making processes, using displacement-based finite elements, the predicted strain energy density variations, stress discontinuities along element boundaries (when the stresses are evaluated directly from the nodal point displacements without stress smoothing) and the magni-

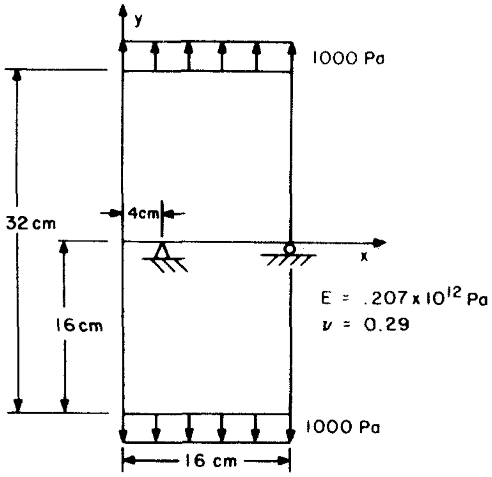


Fig. 11. Analysis of cracked specimen. Analytical value of $K_I = 532$ N/m$^{3/2}$; (a) Specimen considered.

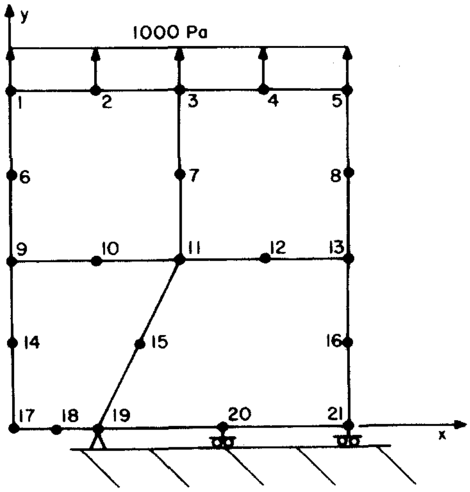


Fig. 11b. Starting nodal point positions; predicted $K_I = 421$ N/m$^{3/2}$.

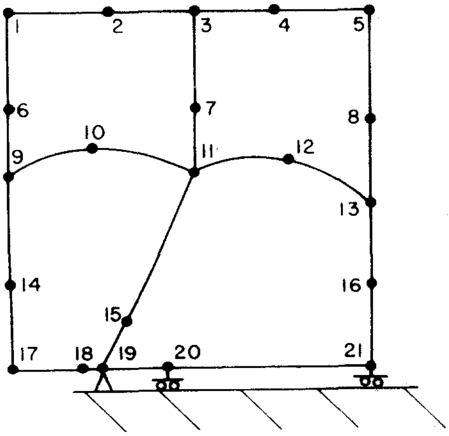


Fig. 11c. Final nodal point positions, predicted $K_I = 523$ N/m$^{3/2}$.

tude of violation of internal element equilibrium can be used. The difficulty in constructing an effective algorithm lies in drawing on the various possibilities and synthesizing a program that in all cases develops an effective mesh. We are at present working to establish such an algorithm in ADINA-MESH.

Fig. 9 shows how a procedure based on element subdivision alone (the h-version refinement process in fig. 8) performed in the analysis of a pressure vessel. In this algorithm the violation of stress continuity between adjacent elements was used to subdivide automatically the elements. It is noted that the final mesh arrived at is not particularly appealing although sufficiently accurate results have been obtained. A considerably more effective mesh can give the same accuracy in the stress prediction [11]. Such mesh is given in fig. 10, where also pressure band plots for the region near the stress concentration are shown. These plots are effective in evaluating the quality of meshes; namely, the bands will be continuous between elements when there are no stress discontinuities.

The procedure of node repositioning is exemplified by the example considered in fig. 11. Here the objective was to calculate the stress intensity factor at the crack tip [12]. A coarse mesh topology was used and the element nodes were repositioned automatically – without the analyst's intervention – to the positions shown in fig. 11(c). Note that in the mesh optimization some mid-side nodal points for the elements at the crack tip were moved automatically to the quarter points, and that the error in the prediction of the stress intensity factor decreased from 21% to 2%. This automatic nodal point repositioning, however, still requires much computation and is currently best used only in certain circumstances, such as special studies of finite element discretizations.

4. Concluding remarks

The objective in this paper was to discuss the use of finite element methods in computer-aided-design. To provide focus to the discussion, emphasis is given to our research and developments in the field and these efforts largely pertain to the development of the ADINA program.

The use of finite element methods is a most important part of CAD and much application is already observed. A large variety of mechanical problems can already be solved and increasingly more complex and nonlinear problems can be tackled because the development of algorithms for such problems is given much emphasis. However, the major impact of finite element methods in the CAD environment will occur when the finite element analysis of a mechanical model – i.e., the choice of finite elements, mesh topology, solution parameters ... – is performed to a higher degree automatically than is current practice. Because of the large potential for using finite elements in such automatic manner, and the corresponding benefits, considerable research and development is currently directed towards this aim. Some ideas in this respect have been briefly discussed in this paper, but the actual realization of detailed algorithms and software for the automatic solution of complex industrial problems will still require much effort over the years to come.

References

[1] K.J. Bathe, Finite Element Procedures in Engineering Analysis (Prentice-Hall, Inc., Englewood Cliffs, New Jersey, 1982).

[2] E.G. Schlechtendahl, Evolutionary aspects of CAE systems, Engineering with Computers 1 (1985) 1–8.

[3] M.S. Shephard, Finite element modeling within an integrated geometric modeling environment: Parts I and II, Engineering with Computers 1 (1985) 61–85.

[4] U. Hellwig, H.-E. Hellwig and M. Paulus, Die Kopplung von CAD und CAM, VDI Zeitschrift 125 (1983), Part I: pp. 355–360, Part II: pp. 455–460.

[5] I. Babuska, E.R. de Arantes e Oliveira and O.C. Zienkiewicz (eds.), Proceedings International Conference on Accuracy Estimates and Adaptive Refinements in Finite Element Computations, Univ. of Lisbon, June 1984.

[6] ADINA Users Manuals, ADINA R&D, Inc., 71 Elton Ave., Watertown, MA 02172, USA.

[7] Proceedings of the 3rd to 5th ADINA Conferences, (K.J. Bathe, ed.):
- Computers and Structures, Vol. 13, No. 5-6 (1981),
- Computers and Structures, Vol. 17, No. 5-6 (1983),
- Computers and Structures, Vol. 21, No. 1-2 (1985).

[8] K.J. Bathe and E.N. Dvorkin, A formulation of general shell elements – The use of mixed interpolation of tensorial components, Int. J. Numer. Meth. in Engrg. 22 (1986) 697–722.

[9] T. Sussman and K.J. Bathe, Finite elements based on mixed interpolation for elastic and inelastic analysis, Computers and Structures, to appear.

[10] K.J. Bathe and E.N. Dvorkin, On the automatic solution of nonlinear finite element equations, J. Computers and Structures 17 No. 5-6 (1983) 871–879.

[11] T. Sussman and K.J. Bathe, Studies of finite element procedures – Stress band plots and the evaluation of finite element meshes, J. Engineering Computations, in press.

[12] T. Sussman and K.J. Bathe, The gradient of the finite element variational indicator with respect to nodal point coordinates: An explicit calculation and applications in fracture mechanics and mesh optimization, Int. J. Numer. Meth. in Engrg. 21 (1985) 763–774.