Engineering Applications of Microcomputers

Edited by
R. F. JONES JR.
K. S. AHLUWALIA
R. S. ROSENBERG

Engineering Applications of Microcomputers

presented at

THE 1986 PRESSURE VESSELS AND PIPING
CONFERENCE AND EXHIBITION AND THE
1986 ASME INTERNATIONAL COMPUTERS IN
ENGINEERING CONFERENCE AND EXHIBITION
CHICAGO, ILLINOIS
JULY 20-24, 1986

sponsored by

THE PRESSURE VESSELS AND PIPING DIVISION, ASME
THE COMPUTER ENGINEERING DIVISION, ASME

edited by

R. F. JONES JR.
DAVID TAYLOR NAVAL SHIP R AND D CENTER
BETHESDA, MARYLAND

K. S. AHLUWALIA
HDS FIBERS INC.
CHARLOTTLESVILLE, VIRGINIA

R. S. ROSENBERG
MICHIGAN STATE UNIVERSITY
EAST LANSING, MICHIGAN

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS
United Engineering Center 345 East 47th Street New York, N.Y. 10017

Library of Congress Catalog Card Number 86-71368

FOREWORD

This volume should be viewed as a progress report on the status of the increasing contribution that microcomputers are making to the engineering profession. A broad spectrum of applications is presented and it is seen that microcomputers are no longer just used for incidential calculations. It appears now that it is time for their exploitation and that they will soon represent a significant part of our capital outlay for computing.

The reason for the rosy picture is the significant progress that has recently been made in the development of machines in the availability of large extended memory, high-capacity hard disks and math coprocessors that are needed for intensive numerical computations such as that required for finite element applications. Also the increasing availability of graphics driven machines, similar to Apple's Macintosh, has brightened the future considerably. These machines are very efficient in terms of requiring less human resources for initial training and routine machine interaction. It appears that the future is virtually unlimited.

The publication represents the joint effort of the Pressure Vessels and Piping and Computer Engineering Divisions. I would like to thank the authors in behalf of both Divisions for making this timely contribution on this important subject. The volume is the material evidence of their hard work.

R. F. Jones Jr.

CONTENTS

ORIGINAL PAGE IS
OF POOR QUALITY

PRELIMINARY DESIGN OF PRESSURE VESSELS AND PIPING SYSTEMS USING THE FEM ANALYSIS ON PERSONAL COMPUTERS

W. K. Ho, R. S. Lahey, and K. D. Blakely
The MacNeal-Schwendler Corporation
Los Angeles, California

ABSTRACT

The technological breakthrough of personal microcomputers has enabled the usage of the Finite Element Method in analyzing structural problems on personal computers. The capability of the microcomputers' hardware, the state-of-the-art FEM software and the relative low cost of microcomputers have made the usage of microcomputers in solving small to medium size engineering problems more effective, costwise and timewise.

This paper shows how the Finite Element Method can be used efficiently on IBM personal computers in preliminary design of pressure vessels and piping systems utilizing the sophisticated, and yet easy to use, MSC/pal 2 program. The application is demonstrated by several examples of static and dynamic stress analysis. The limitations of the computers' hardware and the MSC/pal 2 program, which affect the size of the problems being solved, are also discussed.

INTRODUCTION

The introduction of the Finite Element Method (FEM) more than twenty years ago has proved to be an invaluable way to analyze engineering problems. Originally, this method was used for structural engineering only, performing the static and dynamic elastic analyses. Since then, it has been applied to various applications, such as: heat transfer problems, fluid dynamics, soil foundation and others. However, this method generates an enormous amount of data and performs a lot of calculations that until lately could only be done by mainframe computers. The cost and time for such analyses on mainframe computers are tremendously high and lengthy, respectively. Recently, the introduction of personal computers (PCs) and the ever-improving electronic technology have helped overcome these obstacles.

The feasibility of using Finite Element Method on microcomputers lies on the capabilities and cost of hardware and software. The improvements to the hardware in the microcomputer industry during the past few years make the use of these machines practical. The introduction of new processor chips (such as 8088, 80286 and 8087 chips) increases the speed of performance several times over the older chips. In addition, hard disk drives and floppy disk drives provide an economic and effective way to store data that favors the use of microcomputers. Software has been developed and tailored to meet the needs of users utilizing microcomputers. Furthermore, the cost of microcomputers is much less than the mainframe computers. These gigantic steps in both hardware and software developments, in addition to the convenient access to the machines, make the usage of the Finite Element Method on microcomputer possible.

However, microcomputers are not perfect. Although their speed has improved substantially, they are still slow in solving large problems, such as nuclear power plant and non-linear problems. The Random Access Memory (RAM) size of microcomputers are also restricted by the hardware and software limitations. As a result, microcomputer usage is limited to solving small to medium size problems to achieve time and cost effectiveness. For larger problems, the computing time will be so long that it will be better to perform the analysis on mainframe computers.

A three-dimensional finite element analysis program for the IBM PC XT and AT machines, MSC/pal 2, has been developed by The MacNeal-Schwendler Corporation. The program offers static, normal modes, transient response and frequency response analysis for two- or three-dimensional linear isotropic structures. The program provides a wide variety of powerful and commonly used elements, such as curved and straight beams with tapered or constant-area sections, quadrilateral and triangular plates, shear panel and others, and gives the user a choice to select what he needs. After the analysis has been done, tabular output of nodal displacements, velocities and accelerations, and element forces and stresses can be obtained. In addition, graphical output is available for model geometry, structural deformations, various responses as functions of time and frequency, animated deformation plots and stress contour plots. All these graphical features are valuable tools to isolate errors and help identify the critical areas of the structures at a glance.

MSC/pal 2 can be used on IBM XT and AT personal computers with 512K memory, a floppy disk drive, a hard disk drive, a color/graphics adapter card and a math coprocessor.

In order to optimize the memory space in the computer, a bandwidth minimizer is used in MSC/pal 2 program. Solution algorithms are processed in the main memory to maximize the speed and efficiency. Due to this constraint, the typical problem size is between 500 to 1000 degrees of freedom for both static and dynamic solutions.

As the hardware constraints on the microcomputer limit the capabilities of utilizing finite element analysis to small or medium size problems, some modeling techniques are helpful in increasing the effectiveness of the analysis. A very common modeling technique is symmetry. With the appropriate boundary conditions applied on the model, a full model can be reduced to a half model or even smaller segments; consequently, less memory space is needed. Another technique is the elimination of the less critical degrees of freedom, such as Guyan Reduction (1), for the dynamic solutions, which also increases the effectiveness of using FEM in microcomputers.

This paper illustrates the application of finite element analysis on personal computers, using some modeling techniques on pressure vessels and piping systems. The examples are compared to analytical solutions, experimental data and a different finite element program MSC/NASTRAN (2) (a general purpose FEM program of The MacNeal-Schwendler Corporation for larger computers).

STATIC STRESS ANALYSIS OF CYLINDRICAL VESSEL WITH SPHERICAL CAP UNDER THE ACTION OF INTERNAL PRESSURE

This example compares MSC/pal 2 solution with analytical results (3) for a static stress analysis. A full model of a pressure vessel with spherical caps under internal pressure is shown in Figure 1.

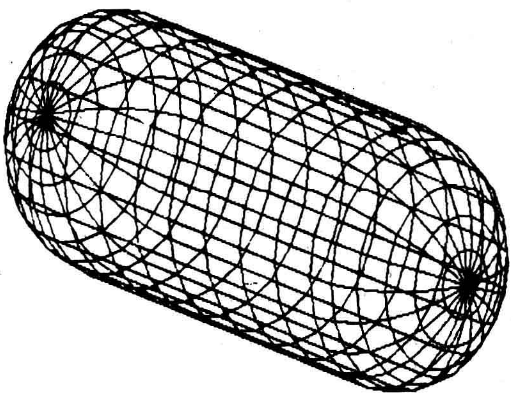

Figure 1. Full model of cylindrical pressure vessel with spherical caps.

The pressure vessel has a constant wall thickness throughout the cylinder and the cap. Due to symmetry of the model and applied loads, only one-eighth of the total structure is considered. The solution of the reduced model adaquately describes the deformed configuration of the whole structure. Figure 2 shows the reduced model.

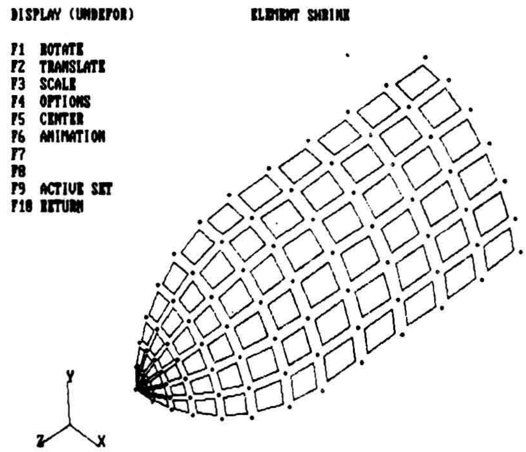

Figure 2. Mathematical model for the pressure vessel with element shrinkage option shown.

An element shrinkage option is invoked in the graphical plot showing the existence of shell elements that idealize the pressure vessel. According to Reference (3), the maximum hoop stress and the axial stress, normalized with respect to the radius of the cylinder, the internal pressure and the thickness of the vessel, are 1.032 and 0.6465, respectively. The computed maximum hoop stress and axial stress by MSC/pal 2 are 1.063 and 0.620, while MSC/NASTRAN's results are 1.019 and 0.633. They agree well with each other. As a general rule, the finer the mesh model, the better the results. In this particular case, the analytical critical stress is not located exactly at the corners or the center of the quadrilateral plate elements, the locations where the stresses are computed. As a result, a small difference in stress is expected. Figure 3 compares the deformed and undeformed shapes of the model. The time required in solving this problem using an IBM AT machine was approximately 7 minutes.

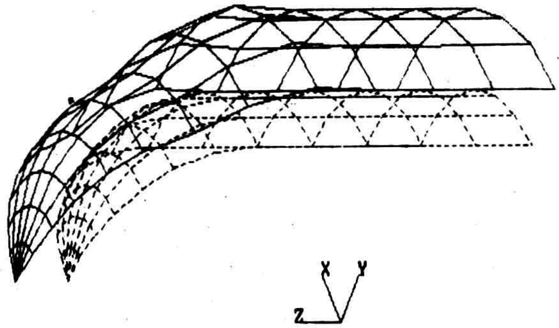

Figure 3. Deformed and undeformed shapes of the pressure vessel under internal pressure.

STATIC STRESS ANALYSIS OF A CYLINDER-TO-CYLINDER INTERSECTION UNDER INTERNAL PRESSURE

This practical example shows the stress analysis of a cylinder-to-cylinder intersection under constant internal pressure using shell elements. In Figure 4 the geometry and physical properties are given. The experimental stress data on the inner and outer surfaces are documented in Reference (4), the computed MSC/pal 2 results are compared to these data. The symmetry of the model also reduces the problem to a quarter model.

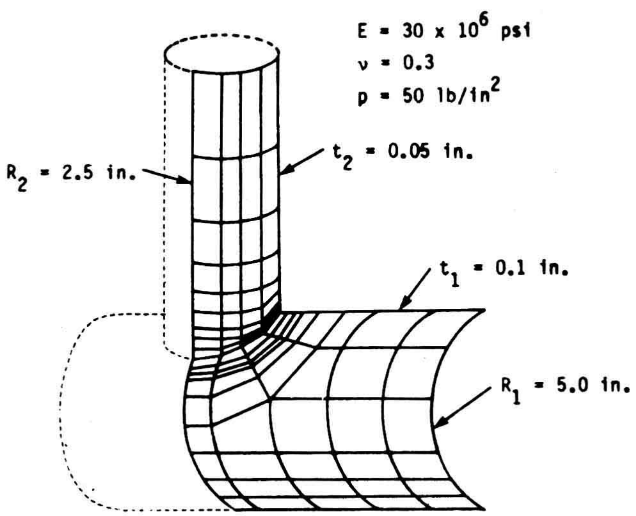

Figure 4. Static stress analysis of a cylinder-to-cylinder intersection subjected to internal pressure.

A good comparison of the stresses is shown in Figures 5 and 6. These figures show that the bending action is severe at the intersection.

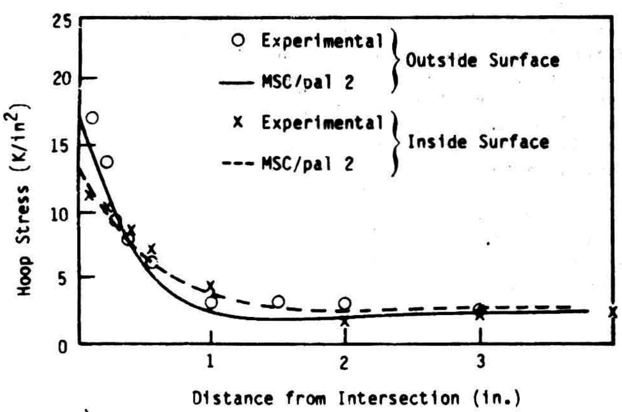

Figure 5. Comparison of hoop stresses between computed and experimental results.

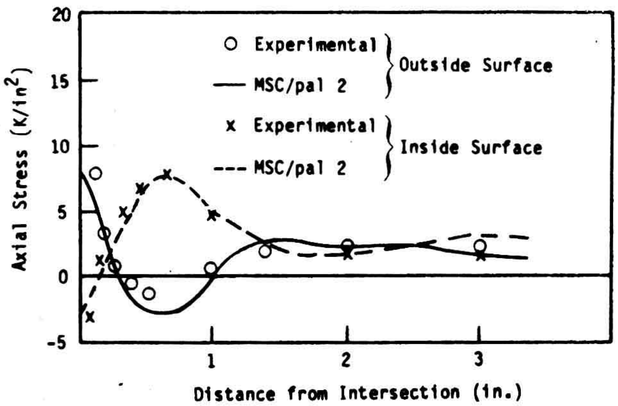

Figure 6. Comparison of axial stresses between computed and experimental results.

As the distance from the intersection increases, the bending effect decreases and membrane stress dominates. A finer mesh at the intersection would give even closer agreement with the test results, due to the fact that this model has a very high stress gradient at this location. Figure 7 illustrates a contour plot of Von-Mises stress with outline option. In addition, a plot showing the element stress values against the elements is shown in Figure 8. The locations of the critical stress are readily determined with this graphical output. Less than 11 minutes was spent to perform the above calculations.

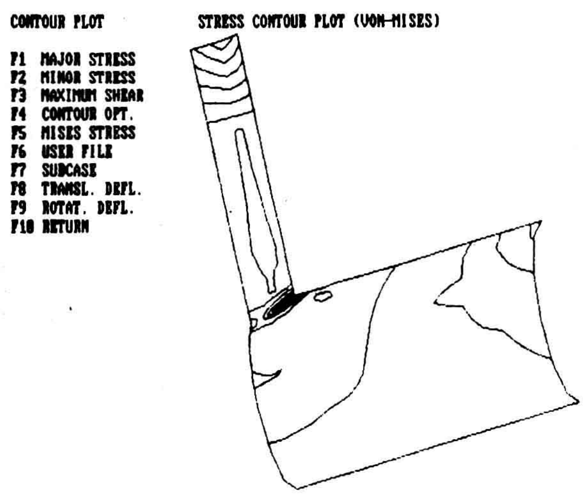

Figure 7. Von-Mises stress contour plot.

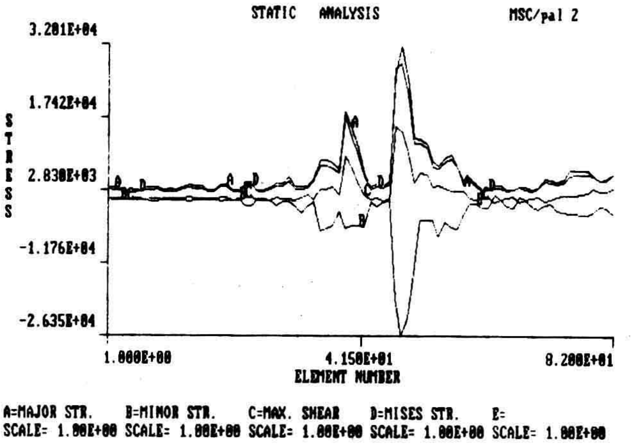

Figure 8. Plot of maximum element stress components vs. element.

SEISMIC RESPONSE OF A PIPING SYSTEM IN A NUCLEAR POWER PLANT

Designs of piping systems in nuclear power containments are not easy or straight-forward. In some situations, even the general purpose FEM program for mainframe computers would have difficulties handling these problems. This example presents an alternate, easy way to perform a preliminary design of a piping structure in a nuclear power containment under seismic actions, which fits well in a microcomputer environment.

For our example, a three-dimensional piping system in a nuclear power containment, which is located at the fifth level of the six level plant, is considered. The dynamic response of the piping system under earthquake loads is investigated. A simple mathematical model is generated for the power containment building using beam elements. Modal analysis is performed on the model to ensure that the few lowest modes are sufficient to represent the actual building. A transient response analysis of the building is performed using a base earthquake excitation. The North-South component ground acceleration time history of the 1940 El Centro earthquake is used as the input for the containment. The acceleration-time response of the fifth floor due to this excitation is then considered as the input acceleration applied to the supports of the piping system. The transient response of the piping structure is analyzed using this excitation. A flow chart showing the process of this whole calculation is summarized in Figure 9, which outlines the sequence of such an analysis.

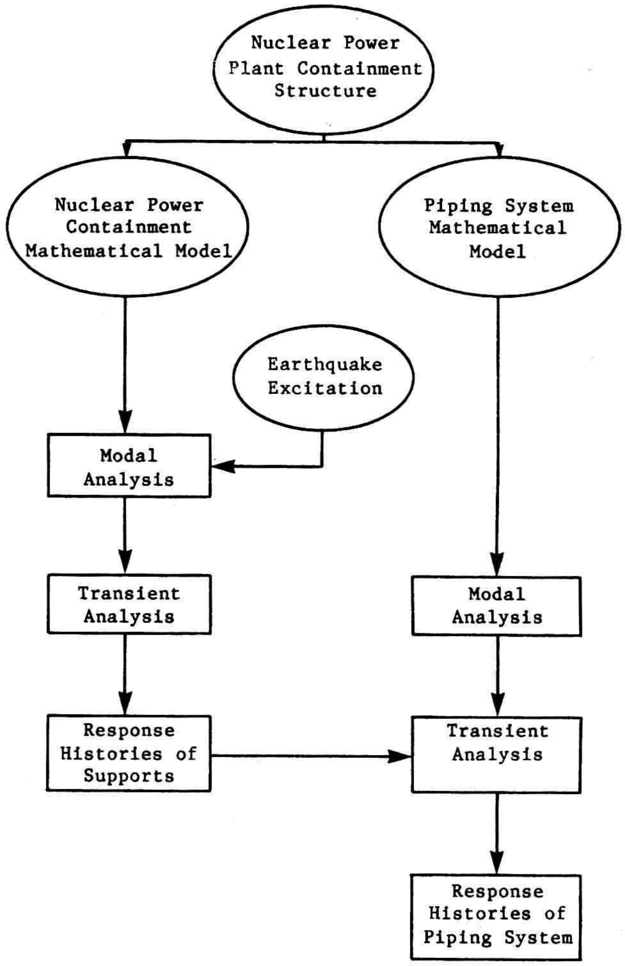

Figure 9. Flow chart showing the transient analysis of a piping system in a nuclear power plant containment.

The following assumptions and simplifications were assumed for this analysis due to various reasons. First, the pipe elbow element is not available in MSC/pal 2. A series of straight beams are used instead to model the bend sections; they are limited to 15° subtended arc angles. Shear effects are also considered. The radial offset of the arc of the neutral axis from the arc of the geometric centroid is significant for the elbow element (5). As the ratio of the radius of curvature to the radius of the pipe increases, the change in radial offset decreases. The ratio is set to two and a half for this example. As a result, the effect in the accuracy of the model is minimized. Second, two to three percent damping ratios, which are typical values for nuclear power plants, are assumed to simulate the actual damping of the structures. Third, as the rotations of the piping as well as some of the translational degrees of freedom are less critical in representing the total structure, they can be eliminated by the method of Guyan Reduction. The elimination is a mean to reduce the number of active degrees of freedom in dynamic analysis with minimum loss of accuracy. The inertia and stiffness properties are redistributed to a smaller but more critical set of nodal points. The dynamic analysis is then performed on this reduced set and the time and cost of the analysis is reduced substantially.

This reduced set of nodal points should be selected evenly along the structure and should include all nodal points with large inertia. For demonstration purposes, all rotational degrees of freedom for all free nodal points and all translational degrees of freedom for alternate free nodal points are eliminated for the piping system.

Figure 10 shows the geometric properties of the three-dimensional piping system. The pipe supports as well as the rigid rod hangers are also shown.

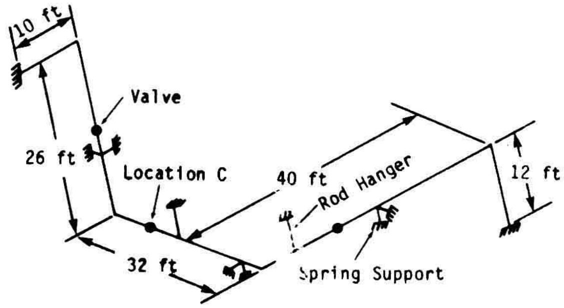

Figure 10. Three-dimensional piping system in a nuclear power containment.

A few concentrated masses are included to simulate the mass of valves. The system is assumed to be filled with water with the mass of the water distributed along the beams. The mode shapes for the first two lowest natural frequencies, 3.1 Hz and 4.0 Hz, are plotted in Figures 11 and 12.

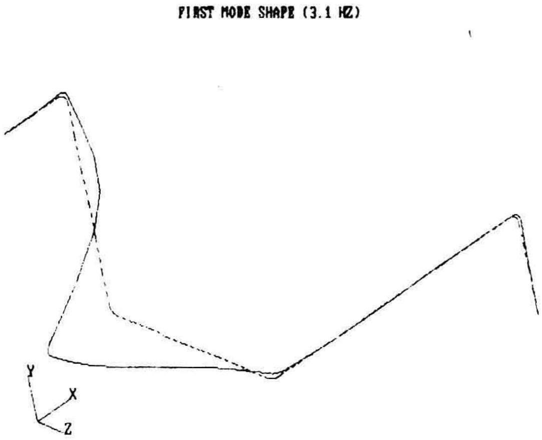

Figure 11. First mode shape of the piping system (3.1 Hz).

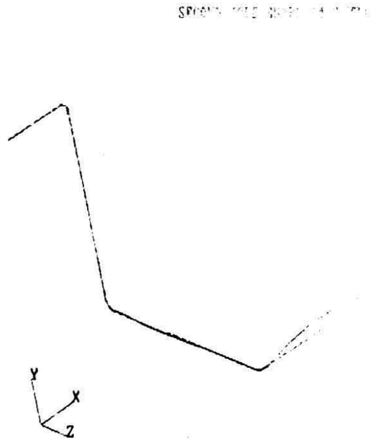

Figure 12. Second Mode shape of the piping system (4.0 Hz).

The El Centro ground acceleration time history and the acceleration response history at the fifth level of the containment are plotted in Figure 13.

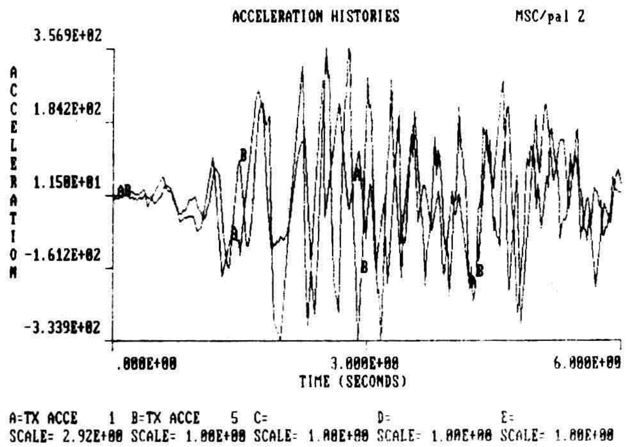

Figure 13. Acceleration excitations for the ground and piping system.

The response acceleration of the power containment is largely amplified during the earthquake, a magnification factor of approximately three is induced at the fifth level. The displacement-time response histories of the floor input and at location C are compared in Figure 14. It can be seen that a magnification factor of 1.5 for the displacement is, likewise, occurring at point C. This transient analysis required approximately 30 minutes computing

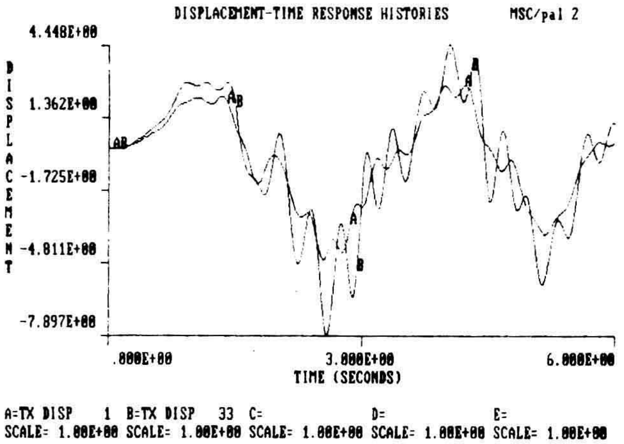

Figure 14. Displacement-time response histories of the piping support and at location C.

This approach is suitable for the preliminary dynamic response studies of piping system under seismic actions, especially for the microcomputer. A more precise model, with the addition of other features such as non-linear effects, can then be analyzed by a more general FEM program using larger computers. MSC/pal 2 also helps to provide this facility by translating the MSC/pal 2 input data into MSC/NASTRAN input format, so that more complex and advanced models can be studied by the use of mainframe computers. This way, the analysis is more efficient timewise and costwise.

CONCLUSION

Various examples of pressure vessels and a piping system under static and dynamic loads are illustrated in this paper by the use of the MSC/pal 2 finite element analysis program for microcomputers. These examples are compared to analytical results and experimental data. Some modeling techniques are also illustrated to reduce the size of the problems so that they can fit into the limitations of microcomputers.

The emergence of personal computers has made the use of finite element analysis more practical for engineers and designers. Even though this development is still in the early stage due to restrictions imposed by hardware capabilities, improvements to microcomputers will make them more attractive in the near future. The desktop accessibility, the relatively low cost of the personal computers and the applicable software are several of the factors that provide greater efficiency to use personal computers as stand-alone machines, work-stations or nodes in a local network to solve moderate size engineering problems.

ACKNOWLEDGEMENTS

We would like to thank Ms. Julie Biggs and Ms. Bita Esfandiari of the MacNeal-Schwendler Corporation for their help in producing an excellent piece of artwork.

REFERENCES

1. Guyan, R. J., "Reduction of Stiffness and Mass Matrices", AIAA Journal, Vol. 3, No. 2, February 1965.

2. MSC/NASTRAN USER'S MANUAL, Version 64, July 1984.

3. Timoshenko, S., and Woinowsky-Krieger S., Theory of Plates and Shells, 2nd Edition, McGraw-Hill Book Company, New York, 1959.

4. Zienkiewics, O. C., The Finite Element Method, 3rd Edition, McGraw-Hill, London, 1977.

5. Schaeffer, H. C., MSC/NASTRAN Primer, Wallace Press Inc., August 1984.

FINITE-ELEMENT FRACTURE ANALYSIS ON A MICROCOMPUTER[1]

J. W. Bryson and B. R. Bass
Oak Ridge National Laboratory
Oak Ridge, Tennessee

ABSTRACT

The ORVIRT.PC microcomputer program allows finite-element fracture analysis on either an IBM PC/AT or PC/XT. When used together with its companion automatic mesh generation program, ORMGEN.PC, a complete fracture analysis consisting of mesh generation, stress analysis, and fracture analysis can typically be performed in 30 to 45 min. ORVIRT.PC is a stand-alone finite-element program with the capability to carry out two-dimensional (2-D) linear thermoelastic stress and fracture-mechanics analyses. It utilizes a virtual crack extension technique that has been modified to include thermal strains. Although this approach reduces identically to the J-integral for 2-D problems, it offers an advantage in application and programming effort over the latter when thermal loadings are considered. Sample problems demonstrate that an accuracy >1% can be obtained with ORVIRT.PC. System requirements are an IBM PC/AT or PC/XT, 512K minimum memory, hard disk, math coprocessor, and IBM Professional Fortran Compiler.

INTRODUCTION

Until recently, finite-element analysis required the computational power of mainframe computers. Elastoplastic finite-element analysis still requires this kind of resource; however, linear elastic finite-element computations can now be performed on certain microcomputers. This paper presents a package of recently developed programs for conducting finite-element fracture analysis on either an IBM PC/AT or PC/XT microcomputer. ORVIRT.PC [Oak Ridge VIRTual crack extension.Personal Computer (1)] is a stand-alone finite-element program capable of performing two-dimensional (2-D) linear thermoelastic stress and fracture-mechanics analyses. A companion program, ORMGEN.PC [Oak Ridge Mesh GENerator.Personal Computer (2)], automatically generates 2-D finite-element models compatible with ORVIRT.PC for subsequent fracture analysis. Either cracked or uncracked geometries may be considered. Eight-noded isoparametric quadrilateral elements are employed everywhere in the modeling, including the crack-tip region. Special crack-tip elements that allow for an inverse square root variation in the near-tip stress and strain fields may be used at the crack tip.

The ORMGEN.PC and ORVIRT.PC microcomputer programs are based to a large extent on techniques used in the mainframe fracture-analysis system ORMGEN/ADINA/ORVIRT (3,4,5) developed under sponsorship of the Heavy-Section Steel Technology Program at Oak Ridge National Laboratory (ORNL). An enhanced version (5,6) of the deLorenzi virtual crack extension development (7) is employed that includes a thermal loading capability. The virtual crack extension technique used here can be shown (5) to reduce identically to the J-integral for 2-D problems. In addition, it offers an advantage in application and programming effort over the latter when thermal loadings are considered, because all integrals to be performed are area integrals, as opposed to a mix of contour and area integrals for the J-integral. Although the technique is applicable in principle to the elastoplastic problem, limitations imposed by the microcomputer environment prevent including an elastoplastic capability at this time.

Comparisons of ORVIRT.PC results with known solutions show agreement within 1%. Typical ORVIRT.PC run times on an IBM PC/AT are 15 to 20 min for a problem approaching the maximum allowable dimensions of 135 isoparametric quadrilateral elements and 450 nodes. The program takes ~60% longer to execute on an IBM PC/XT. A complete ORMGEN.PC/ORVIRT.PC finite-element fracture analysis from mesh generation to the calculation of stress-intensity factors can typically be performed in 30 to 45 min.

[1] Research sponsored by Office of Nuclear Regulatory Research, U.S. Nuclear Regulatory Commission under Interagency Agreements DOE 40-551-75 and 40-552-75 with the U.S. Department of Energy under Contract No. DE-AC05-84OR21400 with the Martin Marietta Energy Systems, Inc.

ORMGEN.PC FINITE-ELEMENT MESH GENERATOR FOR 2-D CRACK GEOMETRIES

Automatic mesh generation is a necessity for fast, efficient finite-element analysis. Mesh generation programs alleviate much of the effort involved in data preparation and eliminate the inevitable data errors that occur during manual preparation of the nodal point coordinates and the element connectivities. Applications of finite-element techniques to fracture-mechanics problems require some additional considerations that make an automatic mesh generation capability all the more important. Often special crack-tip elements are employed to model the appropriate stress singularity along the crack front. A locally refined mesh is usually required in the crack-front region because of the presence of large stress gradients. Finally, certain fracture-analysis techniques may require a special arrangement of elements in the crack-tip region.

ORMGEN.PC automatically generates finite-element meshes for either cracked or uncracked planar geometries. Eight-noded isoparametric quadrilateral elements are employed everywhere in the modeling, including the crack-tip region. Element connectivities and nodal point coordinates are written to an output file in a format compatible for subsequent fracture analysis with ORVIRT.PC.

The program strategy is to divide a structure into a few large blocks or superelements for which the usual manual input of data is required. The fineness of element subdivision within each of these large blocks is specified, and this subdivision is then performed automatically by the program. Because each block has the general form of a quadrilateral, it is convenient to employ the shape functions for eight-noded isoparametric quadrilaterals for locating nodal points via interpolation within a block. Figure 1 demonstrates the mesh generation procedure. Block 1 is a crack-tip block with a predetermined mesh configuration; the remaining three blocks are regular blocks that are subdivided into elements by specifying the number of subdivisions in the local ξ and η directions, respectively. ORMGEN.PC employs modified versions of subroutines (8) for the subdivision procedure. Additional routines provide the appropriate mesh definition at the crack tip, that is, crack-tip block definition.

The outstanding feature of the large-block or superelement approach employed here is its complete generality. ORMGEN.PC can generate a finite-element mesh for almost any conceivable 2-D geometry. Disks, plates, cylinders, and geometries with holes such as the standard compact tension specimen can be readily modeled. Surface or embedded flaws can be analyzed; that is, either one or two crack tips can be included in the model. For cracked geometries, the only restrictions are that the crack lie in the Y = 0 plane and that this plane be a plane of symmetry.

ORVIRT.PC FINITE-ELEMENT FRACTURE-ANALYSIS PROGRAM

ORVIRT.PC is a stand-alone finite-element program capable of performing linear thermoelastic plane stress/strain or axisymmetric analyses. It uses eight-noded isoparametric quadrilateral elements with curved sides and a quadratic variation of displacement. Either a 2 × 2 or a 3 × 3 Gauss integration rule may be employed.

ORVIRT.PC utilizes modified versions of subroutines presented in the finite-element texts by Owen and Fawkes (9) and Hinton and Owen (10). An efficient

ORNL–DWG 86–4183 ETD

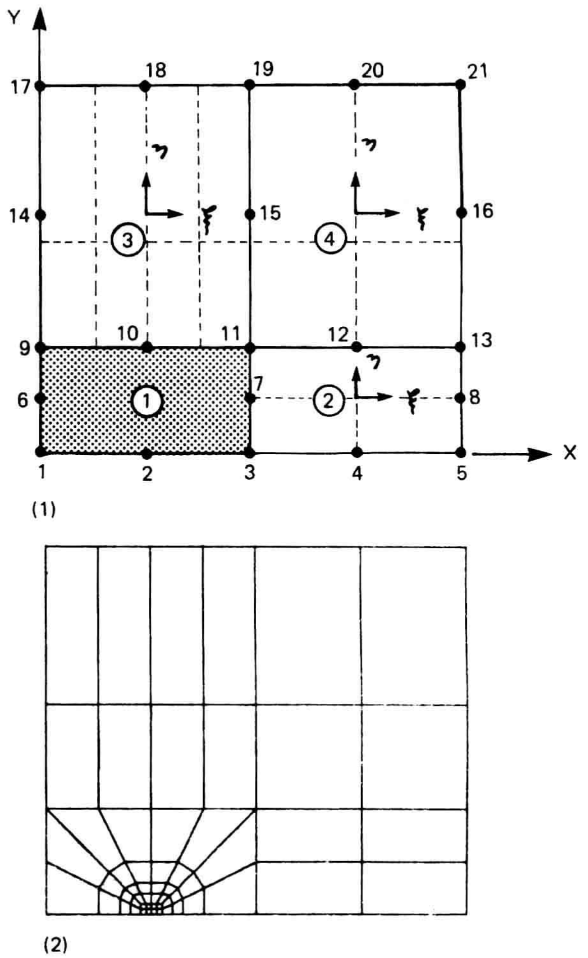

Fig. 1 Finite-element mesh generated by ORMGEN.PC and employed in ORVIRT.PC

(1) Block and block point definition
(2) Element and nodal point definition

frontal solver described and listed in Ref. 9 is used in its entirety in ORVIRT.PC. The frontal technique was originated by Irons (11) and differs from banded solver techniques in that equations are assembled and variables eliminated at the same time. The complete structural stiffness matrix is never formed as such because, after elimination, the reduced equation is immediately written to the hard disk. The frontal technique is therefore particularly suitable for microcomputer applications; both core storage requirements and the total number of arithmetic operations are greatly reduced. Unlike banded solvers, the ordering of elements is very important in the frontal scheme, although the ordering of nodal numbering is irrelevant. Therefore, the elements must be numbered systematically to minimize the front width.

The following types of loading may be considered: (1) point loads, (2) distributed edge loads, and (3) thermal loads. Any combination of these loads may be applied simultaneously, or several loading conditions may be applied in succession to the same structural configuration. Distributed edge loads do not have to be constant but may vary along the element edge. Thermal loading consists of inputting values of ΔT at node points. As a further simplification, gravity loads and other body force loadings are not included.

After deLorenzi (7), ORVIRT.PC employs a virtual crack extension technique that has been modified to include the effects of thermal strains (5,6). The method requires a calculation of the released energy G* corresponding to a virtual crack advance in a cracked body subjected to surface tractions F_α and temperature distribution T (Fig. 2). Points of configuration I (prior to crack advance) are mapped into configuration II (after crack advance) by the mapping

$$\bar{x}_\alpha = x_\alpha + \Delta x_\alpha \ , \qquad (1)$$

where x_α, $\bar{x}_\alpha$ correspond to coordinates in configurations I, II, respectively, and Δx_α is the incremental change in coordinates throughout the body accompanying a virtual crack extension Δa at the crack tip.

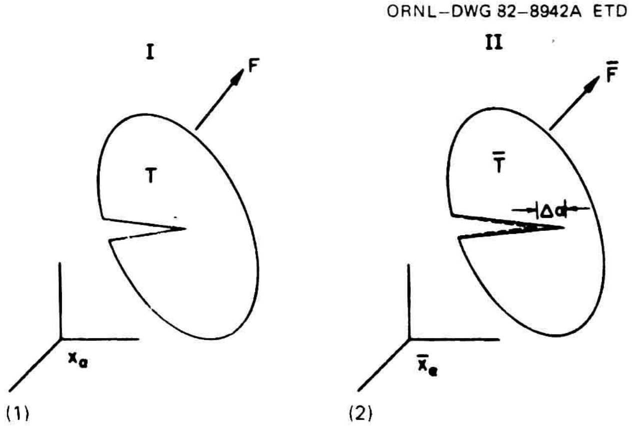

Fig. 2 Crack configuration
(1) Before crack extension
(2) After crack extension

Index notation shows (5,7) that the energy release parameter G* becomes

$$G^* = \int_V \left(\sigma_{\alpha\beta} \frac{\partial u_\alpha}{\partial x_\delta} - W\delta_{\delta\beta}\right) \frac{\partial \Delta x_\delta}{\partial x_\beta} \, dV + \int_V \sigma_{\alpha\beta} \frac{\partial \theta_{\alpha\beta}}{\partial x_\delta} \Delta x_\delta \, dV - \int_S t_\alpha \frac{\partial u_\alpha}{\partial x_\beta} \Delta x_\beta \, dS \ , \qquad (2)$$

where

x_α = spatial coordinates,
V = volume,
S = surface of the cracked body,
u_α = displacement vector,
$\sigma_{\alpha\beta}$ = stress tensor,
W = strain energy density,
t_α = surface traction vector along crack face,
$\theta_{\alpha\beta}$ = strains of free thermal expansion,
$\delta_{\alpha\beta}$ = Kronecker's delta,
Δx_α = mapping function defined above,

and α, β, δ take on the range 1, 2, 3, and the summation convention for repeated indices is used. The strain energy density W is given by

$$W = \int \sigma_{\alpha\beta} \, d\varepsilon'_{\alpha\beta} \ ,$$
$$\varepsilon'_{\alpha\beta} = \varepsilon_{\alpha\beta} - \theta_{\alpha\beta} \ , \qquad (3)$$

where the mechanical strain components $\varepsilon'_{\alpha\beta}$ are defined in terms of the total strains $\varepsilon_{\alpha\beta}$ and the strains of free thermal expansion $\theta_{\alpha\beta}$.

The energy release rate G is given by

$$G = G^*/\Delta A \ , \qquad (4)$$

where ΔA is the area increment covered by the virtual crack extension. The mode I stress-intensity factor K_I is related to G by

$$K_I = \begin{cases} \sqrt{GE} & \text{plane stress} \\ \sqrt{\dfrac{GE}{1 - \nu^2}} & \text{plane strain} \ , \end{cases} \qquad (5)$$

where E is the elastic modulus and ν is Poisson's ratio.

Obviously, an infinite number of mappings Δx_α exist for any given virtual crack extension Δa. However, if a mapping such that $\Delta x_\alpha = 0$ outside a "crack tip zone" is chosen, then the right side of Eq. (2) must be integrated only over this crack-tip zone. In particular, ORVIRT.PC utilizes the mapping shown in Fig. 3, where $0 < \Delta x < \Delta a$ in an outer annulus of elements referred to as Region 1, and $\Delta x = \Delta a$ in Region 2. Note that for 2-D problems, $\Delta x_2 = \Delta x_3 = 0$ and $\Delta x_1 = \Delta x$. The first term on the right-hand side of Eq. (2) is identically zero everywhere except within Region 1, but the second term must be integrated over the entire crack-tip zone, that is, Regions 1 and 2. The third term only comes into play for crack-face pressure loading. ORVIRT.PC constrains a user to model the crack in the Y = 0 plane as shown in Fig. 3; hence, dS becomes dx, and the third term is a straightforward definite integral over the surface of elements within the crack-tip zone (Regions 1 and 2) that have an applied crack-face pressure.

ORVIRT.PC automatically performs the mapping described above; however, a special arrangement of elements is required in the crack-tip zone. This explains why the automatic mesh generator ORMGEN.PC requires a user to model the crack-tip region by choosing from a library of crack-tip block types that have the proper arrangement of elements in this zone.

COMPARISONS WITH KNOWN SOLUTIONS

Two example problems demonstrate the high accuracy obtainable with ORVIRT.PC; each was analyzed using a 3 × 3 Gauss integration rule. From the known solutions, loads and material properties were selected such

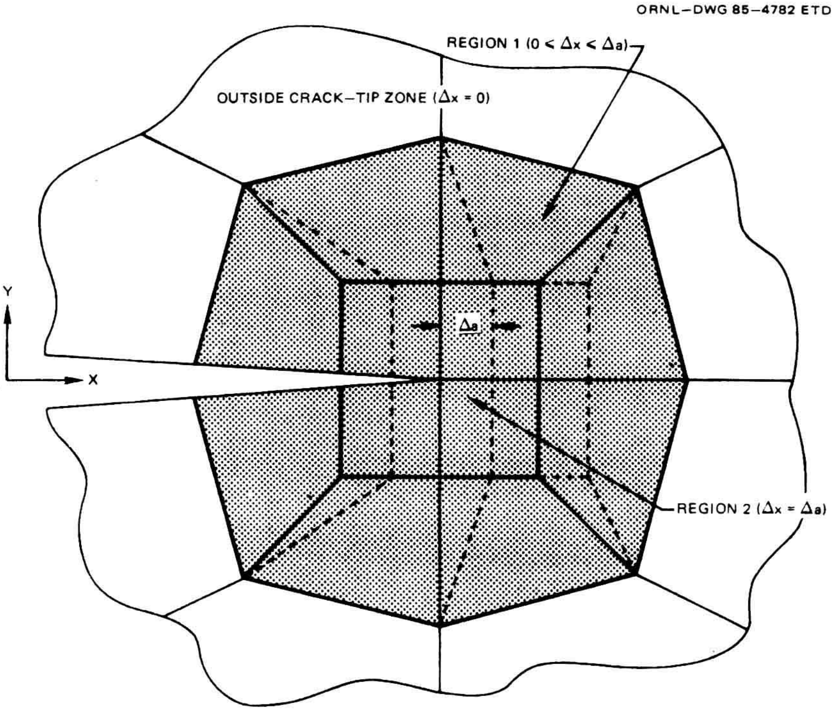

Fig. 3 Crack-tip zone (shaded) over which ORVIRT.PC performs calculations

that K_I = 100;[2] thus percentage differences between ORVIRT.PC results and known solutions are readily obtained.

Standard Compact Tension Specimen, Plane Strain

Figure 4 shows the finite-element model, boundary conditions, and dimensions employed. The known solution (12) for this problem is given by

$$K_I = (P/b)\sqrt{a}\ F_1\ (a/b)\ ,$$

where for $0.3 \leq a/b \leq 0.7$,

$$F_1(a/b) = 29.6 - 185.5(a/b) + 655.7(a/b)^2 - 1017.0(a/b)^3 + 638.9(a/b)^4\ ;$$

for a/b = 0.5, K_I = 100.0 if P = 14.278. Using this load and the given dimensions, ORVIRT.PC computes K_I = 99.9 for a 0.1% deviation from the known solution.

Pressurized Embedded Crack in a Circular Disk, Plane Stress

This example problem demonstrates the capability to model and analyze embedded flaws. Figure 5 shows the mesh generated by ORMGEN.PC as well as the boundary conditions employed in the subsequent ORVIRT.PC analysis. For a/d = 0.5 and d/R = 0.5, the known solution (13) is given by

$$K_I = 1.14p\sqrt{\pi a},\ \text{crack tip A}\ ,$$

$$K_I = 1.20p\sqrt{\pi a},\ \text{crack tip B}\ ;$$

for a = 1, K_I = 100.0 and 105.0 at crack tips A and B, respectively, if p = 49.5. Using this crack-face pressure, ORVIRT.PC calculates 100.6 at crack tip A and 105.8 at crack tip B, representing a 0.8% deviation from the known solution.

PROGRAM OPERATION

The system requirements for executing ORMGEN.PC and ORVIRT.PC are

1. IBM PC/AT or PC/XT, 512K minimum memory, hard disk;
2. math coprocessor;
3. IBM Professional Fortran Compiler, Version 1 or 2; and
4. 132 column printer.

Executing the programs consists of installing the IBM Fortran Compiler routines, copying the program disk containing the source code for ORMGEN.PC and ORVIRT.PC to the hard disk, compiling the programs, preparing input files using an appropriate editor or word processor, and running the programs. ORVIRT.PC writes a complete stress and fracture-mechanics analysis to a single output file.

The programs are written in FORTRAN 77 using 64-bit double-precision, real words. Typical run times

[2]For brevity and ease of preparation, units have been omitted in the following discussion. The results are understood to be applicable for any consistent set of units.

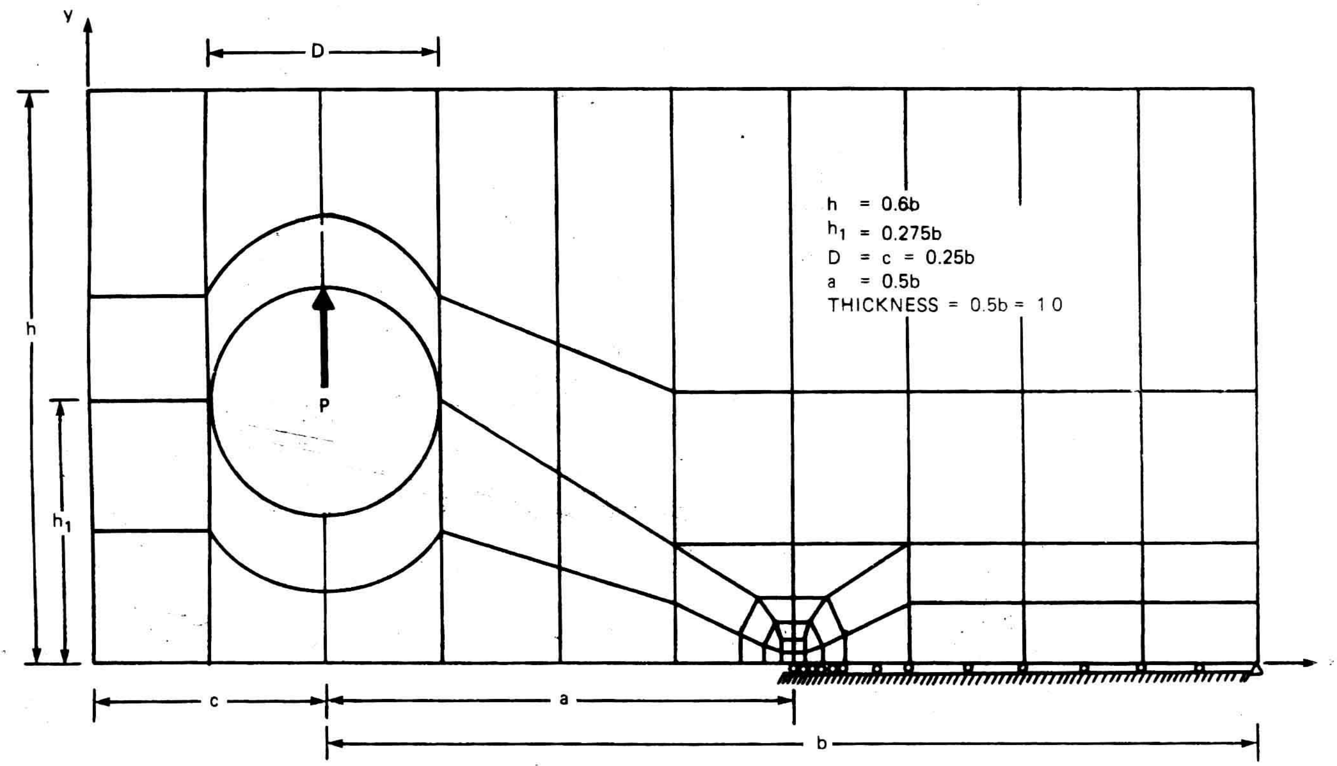

Fig. 4 Finite-element mesh, boundary conditions, and dimensions employed

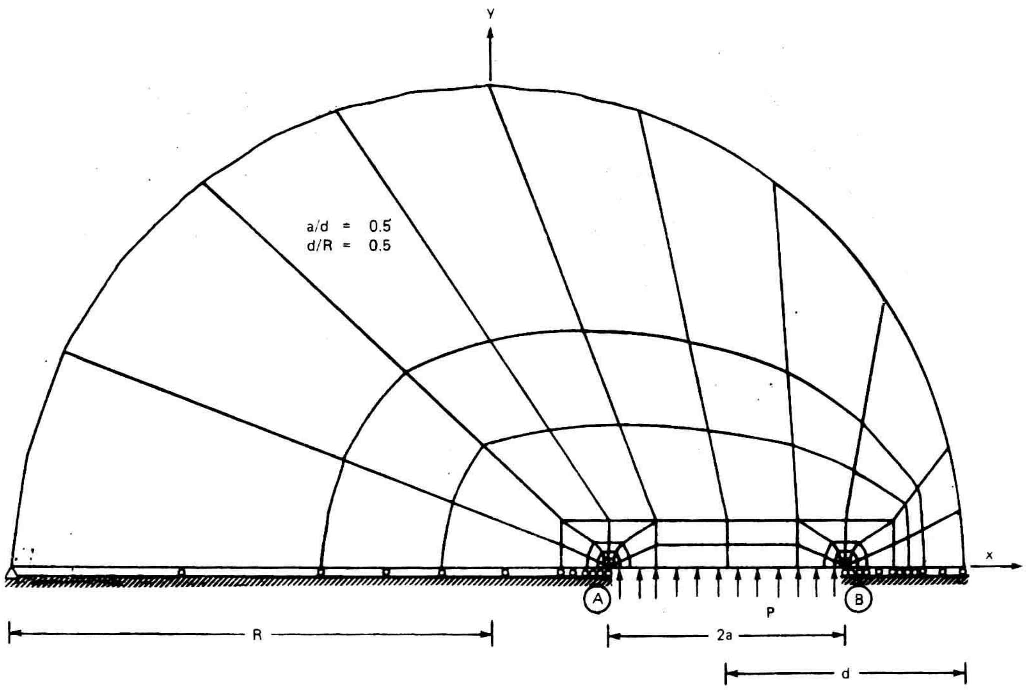

Fig. 5 Finite-element model for pressurized embedded crack in a circular disk