Zeitschrift: IABSE reports = Rapports AIPC = IVBH Berichte

Band: 40 (1982)

Artikel: Adaptive techniques of finite element computations

Autor: Peano, Alberto

DOI: https://doi.org/10.5169/seals-30903

Nutzungsbedingungen

Die ETH-Bibliothek ist die Anbieterin der digitalisierten Zeitschriften auf E-Periodica. Sie besitzt keine Urheberrechte an den Zeitschriften und ist nicht verantwortlich für deren Inhalte. Die Rechte liegen in der Regel bei den Herausgebern beziehungsweise den externen Rechteinhabern. Das Veröffentlichen von Bildern in Print- und Online-Publikationen sowie auf Social Media-Kanälen oder Webseiten ist nur mit vorheriger Genehmigung der Rechteinhaber erlaubt. Mehr erfahren

Conditions d'utilisation

L'ETH Library est le fournisseur des revues numérisées. Elle ne détient aucun droit d'auteur sur les revues et n'est pas responsable de leur contenu. En règle générale, les droits sont détenus par les éditeurs ou les détenteurs de droits externes. La reproduction d'images dans des publications imprimées ou en ligne ainsi que sur des canaux de médias sociaux ou des sites web n'est autorisée qu'avec l'accord préalable des détenteurs des droits. En savoir plus

Terms of use

The ETH Library is the provider of the digitised journals. It does not own any copyrights to the journals and is not responsible for their content. The rights usually lie with the publishers or the external rights holders. Publishing images in print and online publications, as well as on social media channels or websites, is only permitted with the prior consent of the rights holders. Find out more

Download PDF: 11.02.2026

ETH-Bibliothek Zürich, E-Periodica, https://www.e-periodica.ch

Adaptive Techniques of Finite Element Computations

Techniques adaptatives appliquées au calcul par éléments finis

Einwirkung anpassfähiger Verfahren zu Finite-Elemente-Berechnungen

Alberto PEANO
Dr. Sc. Civ. Eng.
ISMES
Bergamo, Italy

Alberto Peano, born in 1946, received his doctoral degree at Washington University, St. Louis, Missouri. He has worked in the industry as a stress analyst as well as in the Polytechnic University of Milano as an assistant in the Structural Engineering Dept. Alberto Peano is head of the Mathematical Modelling Division of ISMES.

SUMMARY
Adaptivity is the capability to increase, either automatically, or with minimal user interaction, the number of degrees of freedom in regions where an error tolerance has been exceeded. The availability of adaptive computer programs will drastically simplify preprocessing of finite element analyses and is particulary useful for three-dimensional application. A benchmark problem is also presented.

RESUME
La notion d'adaptatif s'applique à la possibilité d'ajuster, automatiquement ou sous l'effet d'une intervention minimale de l'utilisateur, le nombre de degrés de libertés à la précision souhaitée dans les zones où les tolérances ne sont pas observées. La disponibilité de tels programmes apportera une simplification considérable au niveau des pré- et post-processeurs, notamment pour les applications tridimensionnelles. Un exemple est exposé à titre de comparaison.

ZUSAMMENFASSUNG
Mit Anpassbarkeit meint man die Fähigkeit, automatisch oder mit geringster Benutzereinmischung die Zahl der Freiheitsgrade dort zu erhöhen, wo eine vorgeschriebene Fehlertoleranz überschritten wird. Die Verfügbarkeit anpassfähiger Computerprogramme wird die Vor- und Nachlaufarbeit von Finite-Elemente-Analysen sehr erleichtern und kann mit Vorteil bei dreidimensionalen Berechnungen angewandt werden. Ein Benchmark-Test wird dargestellt.

1. INTRODUCTION

Analysis of solid continua by current general purpose programs is a very time consuming and expensive task. Unless geometry is favourable to automated mesh generation, preparation of the model may turn out to be a long and tedious chore. Of course the use of advanced graphic capabilities is very helpful. Since estimates of time and cost required for the preparation of a solid model are sometimes excessively optimistic, perspective users may wonder whether results will be available in due time and may select alternative simplified models, if applicable (2D continuum or thick shell).

Moreover the results of expensive 3D analyses are sometimes of little use because of the difficulty of interpreting and gaining confidence in the computed stress values. Two apposite extreme situations may occur. On one hand the user may be disappointed by severe stress discontinuities occurring at element interfaces of coarse meshes. On the other hand the user may be overwhelmed by the huge amount of output data.

Since the cost of data processing has decreased considerably in recent years, it is possible to envisage a time when solid finite element models will be as usual as 2D models have been in the seventies. The previous discussion indicates that before this time comes, two conditions have to be met: model preparation must be fast and simple, result interpretation must be as much automated as possible.

It is well recognized that sophisticated graphic capabilities are essential for an efficient solution of both problems. Graphics alone, however, cannot achieve the ultimate goal of making 3D analysis as simple as 2D analysis is today. Refined 3D meshes are difficult to scrutinize, if at all, on a graphic terminal. A major revolution of the basic finite element techniques is needed and it is effectively under way. Two aspects are particularly relevant to this discussion, namely p-convergence and adaptivity.

P-convergence is a new efficient convergence process that has been investigated in theoretical studies, tested in numerical experimentations and applied to the solution of large application problems. The distinguishing feature of this new convergence process is that the number and distribution of finite elements are fixed, and the number of interpolating (basis) functions, which are complete polynomials of order p, is progressively increased over each element. On the other hand, in the conventional convergence process, called <u>h-convergence</u>, the number and type of basis functions are fixed, and the finite element mesh is refined in such a way that the maximum diameter of the elements, <u>h</u>, approaches zero.

It has been found that in general the rate of p-convergence is faster than the rate of h-convergence (unless optimally graded sequences of mesh refinements are used). The fast convergence rates provided by the p-version of the finite elment method may obviously lead to considerable savings of computer time. A different point is stressed here, however: the p-version may lead to considerable savings of manpower because very coarse meshes may be used. Coarse meshes are important both in data preparation and during assessment of the accuracy of computed results. The main reason is that computer graphics is simplified tremedously by the use of coarse meshes.

For istance displaying results on a plane arbitrarily cutting through the solid has a cost (and a response time) which increases very rapidly with the number of elements of the mesh. Moreover coarse meshes lead naturally to the "hidden variables" concept, which, as discussed later in the paper, is essential in order to keep the computer print-out to a manageable size.

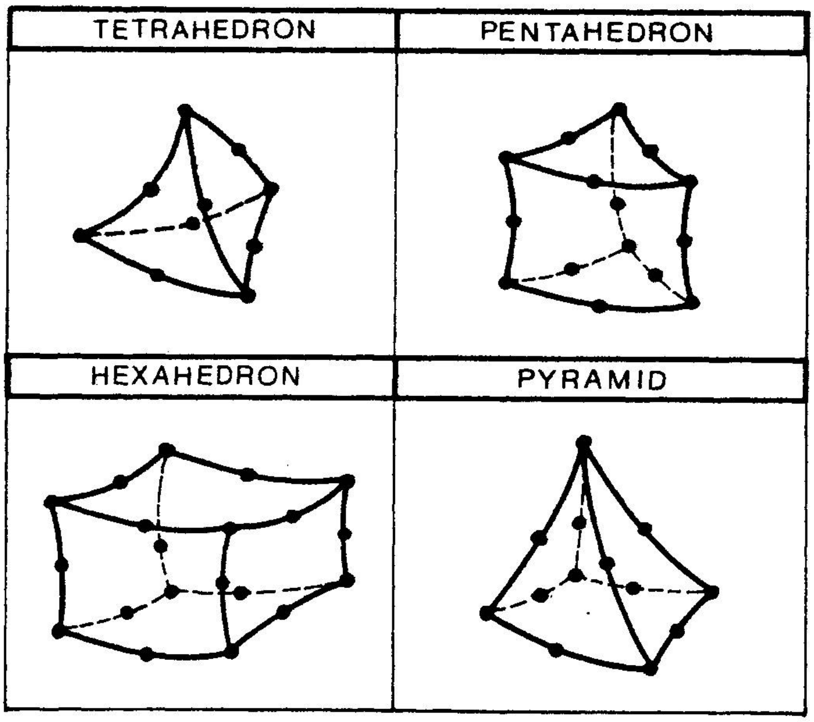
The second fundamental aspect, mentioned earlier, is adaptivity. A finite element software system is said to be adaptive when it possesses some local a-posteriori error estimation capability and a capability to increase, either automatically, or with minimal user interaction, the number of degrees of freedom in regions where an error tolerance has been exceeded. Adaptiveness aims to minimizing the role of the so called "engineering intuition" during post processing of finite element solutions.

Adaptiveness based on h-convergence has been examined by Babuska and Rheinboldt (1). Adaptiveness base on p-convergence has been studied both by B. Szabo (2) and by the author (3, 4). Completely automated adaptive finite element analysis of 2D continua using p-convergence has been tested at ISMES since 1977 (5). The development of a similar capability for 3D analysis has been delayed by the need of performing extensive research and of implementing a strong 3D software system based on the p-version. This software has now been released for commercial use, under the trademark FIESTA.

The theory used to develop FIESTA has been presented earlier (6). In this paper the capabilities and some software design aspects of FIESTA are discussed. The paper also shows results of a benchmark test which was solved both using FIESTA and NASTRAN.

TABLE I

Solid finite elements available in COMET-3D



2. MODELING CAPABILITIES OF FIESTA

The distinguishing feature of FIESTA is the capability of grading the degree of polynomial interpolation over the mesh. User's are allowed to selectively increase the order of approximation over one or more elements without any modification of the stored input data. The maximum degree of interpolation is a complete quartic polynomial.

The first release of FIESTA may be applied to the static stress analysis of three-dimensional elastic continua. Isotropic, transversally isotropic and generally anisotropic material properties are accounted for. The element library is comprised of hexahedra, pentahedra, tetrahedra and pyramids (Table I).

The large variety of element shapes available is very useful for modeling complex details and for mesh grading. All elements have a variable number of nodes.

A large variety of load types may be handled by the program: forces, pressures (e.g. hydrostatic load), general traction load, gravity loading, thermal strains, initial stresses. Any number of load cases and load combinations may be considered.

FIESTA incorporates strong graphic capabilities as well. The mesh and the deflected mesh may be displayed (hidden line option, available). Contours of various displacement or stress components may be plotted on any user' specified surface or segment.

3. THE "HIDDEN VARIABLES" SOFTWARE DESIGN CONCEPT

A characteristic feature of conventional finite element techniques is the simple physical meaning of the unknowns (nodal variables) used for parametric modeling of the structural behaviour. For instance the unknowns used for stress analysis of continua are generally displacements of the same nodal points used for description of the initial geometry. Hence it is quite natural for program developers to provide printed output at all the nodal points used in the model. This feature is very practical and user oriented until the number of degrees of freedom is limited. For refined meshes and large scale analysis problems the situation is completely different:

a) Printed information is redundant because very close displacement or stress values are printed at adjacent nodes.

b) Printed information is excessive. The few meaningful values are not easily accessible in a print-out full of unnecessary information.

c) The conventional choice of the problem unknowns is a well known cause of ill-conditioning in the presence of very refined meshes.

The development of adaptive programs based on h-convergence would actually worsen the problem, as many new undesired nodes are added at locations which are not meaningful for the analyst. In practical applications a compact print-out is always very useful for getting an overall impression of the solution. This is particularly important for solid continua because graphics will exhibit results only on surfaces or sections, which are best chosen after a quick look to the overall solution. For this purpose the analyst would like to have printed output only at a limited number of nodes.

This task is naturally achieved by the p-version of the finite element method. Here very coarse discretizations may be used. In FIESTA the geometric description of the solid is achieved by using standard isoparametric elements, such as hexahedra or pentahedra with straight or parabolic edges. The vertex and midside nodes of this mesh are limited in number and are distributed all over the structure including any region meaningful to the stress analyst.

The natural choice is to provide printed output at these nodes only, irrespective of the selected order of polynomial approximation. The actual degrees of freedom of FIESTA are the vertex node displacements and the amplitudes of hierarchic higher order deformation modes added in order to reach the user specified degree of polynomial approximation. Typical shape functions used by FIESTA are shown in Figure 1. All these additional degrees of freedom are hidden to the user in order to contain the amount of printed information and to simplify the comparison of results computed for different levels of approximation.

Of course results may be printed by the program, upon user request, at any point arbitrarily located in the structure. Moreover FIESTA has the capability of intersecting the solid with an arbitrarily fine 2D grid, to be used for graphic output (contours of any displacement, strain or stress component). Of course the numerical values at the nodes of the auxiliary grid may be printed out.

In conclusion the amount of numerical data printed by FIESTA may be as limited or as large as desired and it is completely independent from the number of elements and of degrees of freedom used to model the problem.

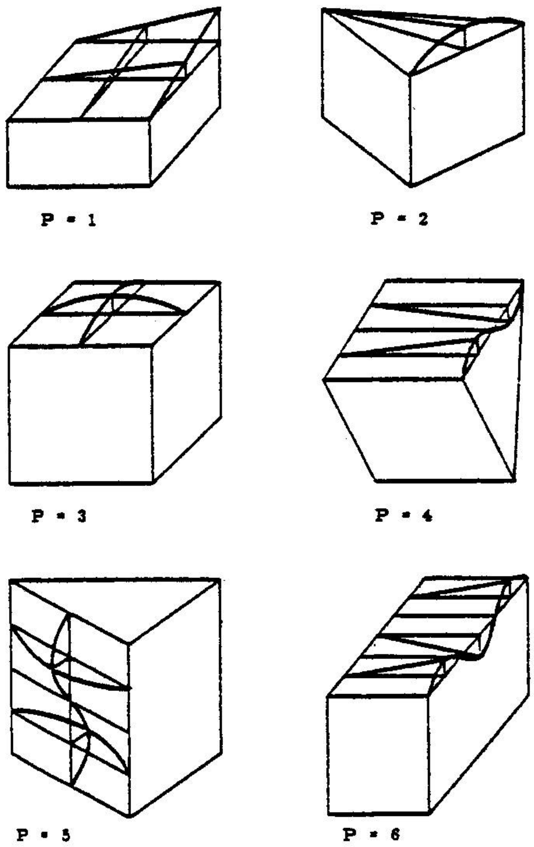


Fig. 1 - Typical shape functions used by COMET for various levels of interpolation.

4. QUALITY ASSURANCE AND INTERACTIVE MODE PROCESSING

FIESTA has been designed as a system of independent processors which operate on a common data structure. Each processor has a specific function, easily recognized by the engineer-analyst. A list of the most important processor is given in Table II. Additional processors and utilities are available for checking the data and for plotting graphs, contours and the deflected grid.

The processor structure is very important for reasons of maintenability and verificability of software. Moreover considerable advantages are accrued in case of interactive and distributed processing. Infact the various processors may be executed during different jobs. This allows several individuals to work independently during data preparation or output of graphic results in order to meet pressing deadlines. Moreover the most CPU consuming tasks may be processed by the most powerful computers of the net (in case of distributed processing) or may be scheduled during night shifts (particularly when working on large minicomputers such as VAX-11/780). An additional advantage is that printed output is generated in steps according to the user needs. For istance nodal stresses may be output after checking displacements or even after plotting stress contours on appropriate sections.

TABLE II

Typical processor functions in FIESTA

PROCESSOR	FUNCTION
TOP	Initialization and input of the grid
PROP	Input of a library of material properties
CONST	Input of Constraints
LOADS	Input of Loading data
PLEVEL	Input of element order of interpolation
LOVE	Computation of the Load Vectors
LCOMB	Input of load combinations
ARRAY	Computation of arrays needed for element matrices
STIFF	Computation of element stiffness matrices
STATIC	Matrix factorization using the frontal technique
SOLVE	Solution of the system of linear equations
DISP	Output of nodal displacements and reactions
STRESS	Output of nodal stresses
FDATA	Output on segments intersecting the grid
CMESH	Generates auxiliary 2D meshes for contour plotting
CDATA	Output at nodes of auxiliary 2D mesh

The order of execution of processors is very flexible provided a meaningful sequence is followed. In case of reanalysis, processor functions, which are not influenced by the modification of the input data, don't have to be executed again. For instance the solution of a new load case does not require triangularization of the stiffness matrix, or a change in the constraints does not imply that the element stiffness matrices have to be computed again.

A reason of concern is that the exceptional flexibility of FIESTA may ease misuse. Infact in case of subsequent modifications (e.g. new material properties) or new load cases the data sets generated by the project may become not only numerous but ambiguous as well. For instance identical files containing different version of the triagulated stiffness matrix may be available in the application data base. In order to eliminate any reason of concern, which could be very serious for users working in a quality assurance environement, an innovative book-keeping system has been built into the program. All runs of a specific project are assigned their ordinal number which identifies the computer print-out as well. The same number as well as additional information (passwords, problems titles, etc.) is used to label the data sets. In this way the program may keep a record of previous execution and issue a Project Status Bulletin at the beginning and at the end of each Computer Report. Therefore the history of any FIESTA output data is fully traceable.

5. A BENCHMARK PROBLEM

The problem was a gear case that was cracking in an area near the gear case mounting bolts. The purpose of finite element analysis of existing design was to obtain a model that predicted the existing failure zones, then this model could be modified to evaluate proposed design modifications prior to manufacturing them. This study was conducted at MCAUTO (McDonnell Douglas Automation) using standard general purpose programs. Later part of it was duplicated using FIESTA for comparison purposes. A plate finite element model was made of the gear case, gear-bearing support structure and support structure. The plate elements did not model the complex gear case geometry in the area of failure. This area contained reinforcing bosses and gussets. Therefore, a local refined model using solid finite elements was required to obtain the local stresses. The loading for this refined model would be the boundary displacements obtained from the plate finite element model analysis.

5.1 The NASTRAN solution

The 8 node CHEXA solid finite elements and the 6 node CPENTA solid wedge elments from MSC/NASTRAN were used for the refined model.

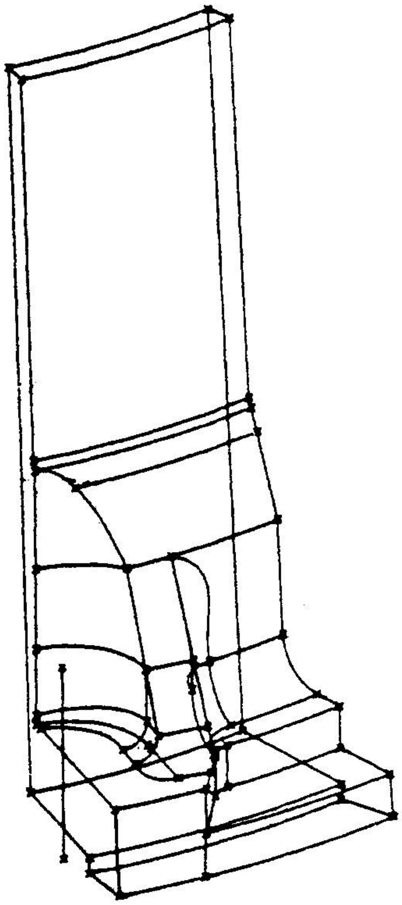


Fig. 2a - CADD model of the gear-case detail.

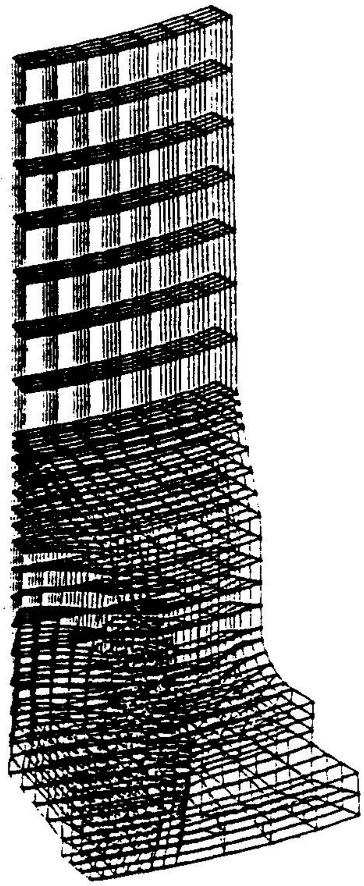


Fig. 2b - NASTRAN model.

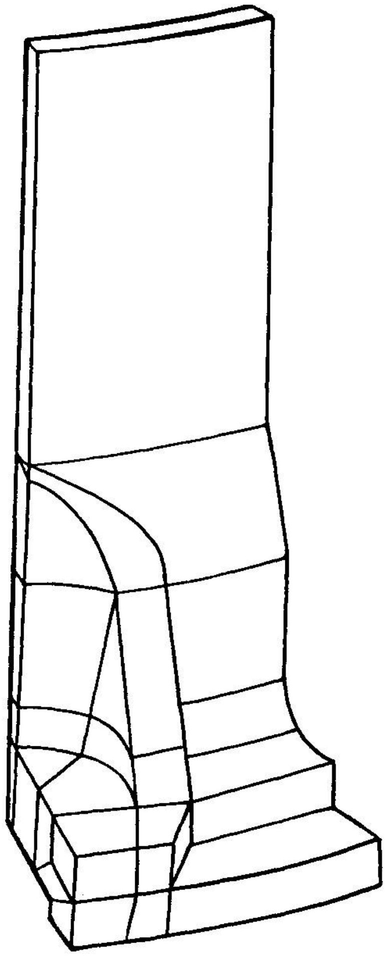


Fig. 2c - FIESTA model.

The geometry of the gear case is rather complex. The outer surfaces of the refined model which consist of a portion of an 18° segment of the gear case was modeled using MCAUTO's Computer Aided Design Drafting System (CADD) and is shown in Figure 2a. The gusset and bosses make this a very complex shape. These geometric data was then passed to MCAUTO's FASTDRAW/3system which is an interactive graphycs finite element modeling system. FASTDRAW was used to divide this geometric model into ten volumes. The volume mesh generation was used to create the finite element model shown in Figure 2b. This model consisted of 1,788 nodes, 1,219 8-node and 35 6-node elements. The number of equation to be solved was 5,188, with an RMS wavefront of 195 degrees of freedom.

Stress results were post processed using MSGSTRESS and plots of the invariant stresses were made. Fig.3, the major principal stress, shows the highest tensile stress in the exact areas that were cracking.

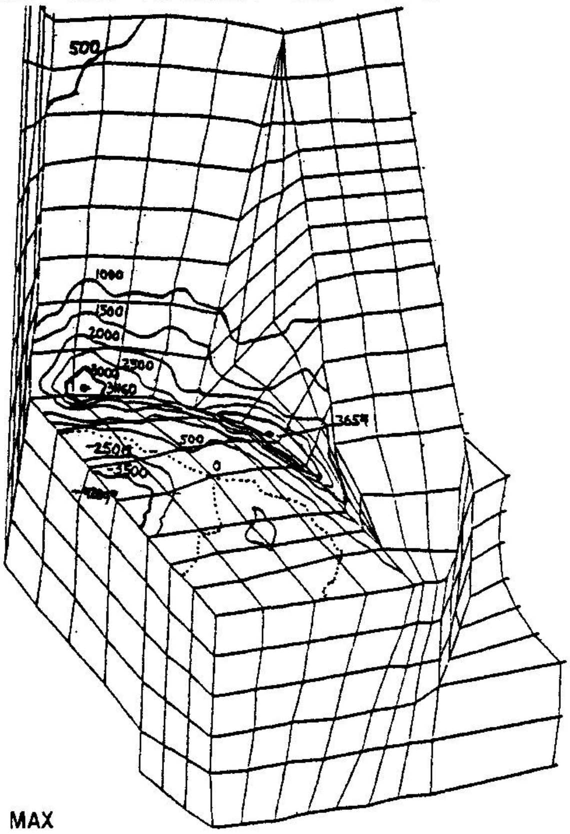


Fig. 3 - Contours of first principal stress in the area of cracking.

5.2 The FIESTA solution

The FIESTA grid shown in Figure 2c is comprised of 28 elements and 222 nodes. The problem was solved using three different levels of interpolation: quadratic, cubic and quartic. The corrisponding number of degrees of freedom is reported in table III.

TABLE III

	NASTRAN	FIESTA		
NUMBER OF NODES	1788	222		
NUMBER OF ELEMENTS	1254	28		
		pl=2	pl=4	pl=6
NUMBER OF DEGREES OF FREEDOM	5188	666	1473	2625
MAXIMUM WAVEFRONT	195	81	149	236

A detail of the contours of the first principal stress is shown in Figure 4. It may be noted that:

- The regions of stress concentration computed by FIESTA are in agreement with both NASTRAN results and experimental areas of failure.
- The discrepancy in the peak stress values is due to the mathematical singularity occurring at the reentrant edge. Infact the theoretical elastic solution exhibits infinite stresses along the edge and higher and higher stresses

may be computed by providing additional degrees of freedom near the irregularity. Realistic values of stresses, if needed, should be computed by elastic plastic analysis and require to model the real radius of curvature of the irregularity (compare fig. 2aand fig. 2c).

- The regions of probable failure are clearly pointed out by the quadratic solution already. The cubic solution is needed just to confirm the results. The combined computer cost of a quadratic and a cubic solution is obviously less than the cost of the NASTRAN solution. Note that quadratic elements are available in NASTRAN as well. Coarse meshes are difficult to use, however, because the accuracy of the results is not easily assess in NASTRAN.

- The stress oscillation between FIESTA solution are due to the singularities at the reentrant edge and in the area where the bolt load is applied. This is a very large concentrated force which causes a singularity of the Boussinesq type. FIESTA provides the best energy-approximation, although it is unable to interpolate the singular solution by standard polynomials. This is not a reason of concern, as the mathematical irregularities have no engineering significance. NASTRAN provides smoother but equally unrealistic stress results in the singular regions.

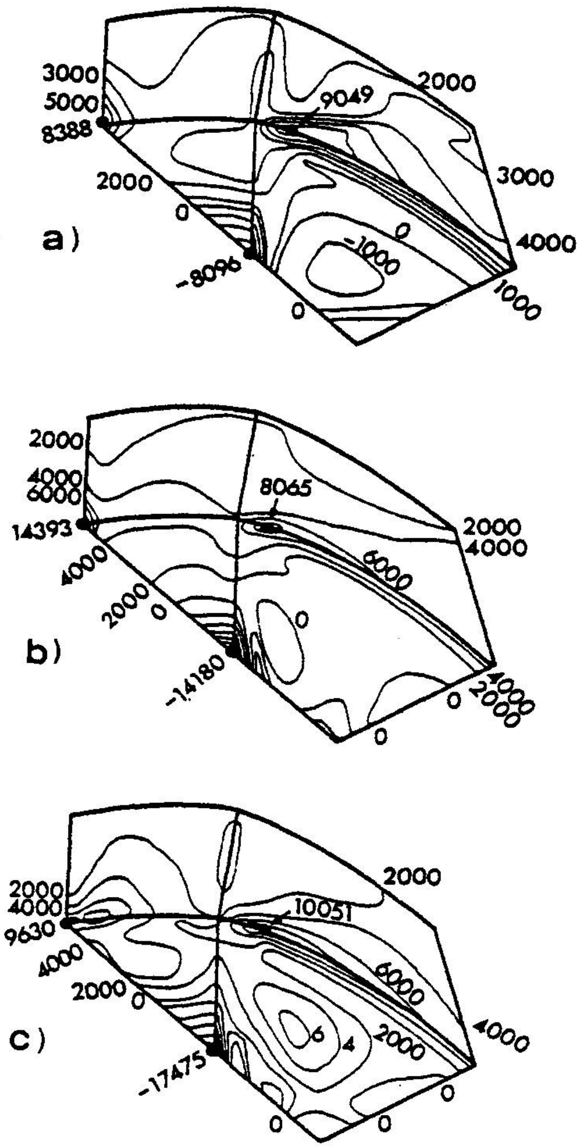


Fig. 4 - Contours of first principal stress for a) quadratic, b) cubic and c) quartic polynomial interpolation.

6. CONCLUSIONS

A new software system for the modeling of solid continua has been released for commercial use. The new system is based on recent theoretical advances and is sensibly different from state-of-the-art finite element programs. An example of successful application of FIESTA has been reported. It is hoped that the new system will get a favourable acceptance by the finite element community. Extensive use of FIESTA is necessary to assess the merits of the new approach.

ACKNOWLEDGEMENTS

The NASTRAN solution was computed by Mr. James Walker, Principal Consultant for Structural Engineering, McDonnell Douglas Automation (MCAUTO). Sincere thanks for suggesting this exacting benchmark problem.

REFERENCES

1. BABUSKA I., RHEINBOLDT W.C., A-Posteriori Error Estimates for the Finite Element Method. Tech. Rep. TR - 581, University of Maryland, Sept. 1977.
2. SZABO B., BASU P.K., ROSSOW M.P., Adaptive Finite Element Analysis based on P-convergence. NASA Conference Publication 2059, pp. 43-50, Nov. 1978.
3. PEANO A., FANELLI M., RICCIONI R., SARDELLA L., Self-Adaptive Convergence at the Crack Tip of a Dam Buttress. Proc. Int. Conf. on Num. Meth. in Fracture Mechanics, Swansea U.K./9-13 Jan. 1978.
4. PEANO A., RICCIONI R., Automated Discretization Error Control in Finite Element Analysis. 2nd World Congress on Finite Element Method, Bournemouth, U.K., 23-27 Oct. 1978.
5. PEANO A., PASINI A., RICCIONI R., SARDELLA L., Self-Adaptive Finite Element Analysis. Proc. VI Int. Finite Element Congress, Baden Baden, Nov. 1977.
6. PEANO A., PASINI A., RICCIONI R., SARDELLA L., Adaptive Approximations in Finite Element Structural Analysis. Computers and Structures, Vol. X, pp. 333-342, 1979.