The MacNeal-Schwendler Corporation
1990 MSC World Users Conference
Los Angeles, California

MSC/PROBE: AN OVERVIEW

Barna A. Szabó

Albert P. and Blanche Y. Greensfelder Professor of Mechanics
and Director, Center for Computational Mechanics
Washington University, St. Louis, MO 63130

ABSTRACT

The technological base of MSC/PROBE is discussed from the point of view reliability and quality assurance in the engineering decision-making process. Some aspects of the computer implementation are discussed and an example is presented.

INTRODUCTION

The technological base of MSC/PROBE, usually referred to as the p-version of the finite element method, fully matured only in the mid-1980's. The year 1984 brought many important new developments and perspectives, and 1984 marks the reaching of maturity in a business sense as well: Noetic Technologies Corporation, the company which broke ground for the new finite element technology in the engineering software business, was formed in November 1984, and was acquired by The MacNeal-Schwendler Corporation in December, 1989. This acquisition signalled the acceptance of this technology by the largest and most important organization in the finite element software business. Acceptance occurred gradually in the academic and research communities between 1981 and 1989. The growing interest of these groups is evidenced, for example, by a forthcoming special edition of the journal: Computers in Applied Mechanics and Engineering, in which the papers presented at the Workshop on Reliability in Engineering Computations, held in Lakeway, Texas in October of 1989 will be published. The following paragraph is from the preface to this edition which was drafted and discussed at the Workshop:

> *"The possibility of delivering exponential convergence rates in practical engineering simulation is very significant; if achieved, as results presented at this Workshop clearly show that such convergence rates are not uncommon in p- and hp-methods, then these approaches may emerge as the most important modelling methods available. In principle, such methods can give results of a specified accuracy on a machine with fixed memory that cannot be attained by any conventional (low order, unadapted) method."*

The key participants in the development of the algorithmic processes on which MSC/PROBE is based were the researchers in two academic institutions, the Center of Computational Mechanics at Washington University in St. Louis and the

Laboratory for Numerical Analysis at the University of Maryland; the investors; the professional and business staff, and the end-users. The importance of a healthy and well functioning relationship among these groups, and smooth and effective collaboration among the members of a balanced development team in which the disciplines of applied mathematics, engineering and computer science are well represented, cannot be overstated.

In this paper the question: 'Why is the p-version of the finite element method important?' is addressed from the point of view of reliability and quality assurance in the engineering decision-making process.

FUNDAMENTALS AND NOTATION

Corresponding to any well formulated mathematical model is an exact solution, denoted by $\vec{u}_{EX}$, which depends on the choice of formulation and the data which characterize the geometry, material properties, loading, and constraints, but is independent of the discretization. Corresponding to a particular choice of discretization is a finite element solution, $\vec{u}_{FE}$, and a number N, called the number of degrees of freedom, which is the number of linear algebraic equations one has to solve in order to obtain $\vec{u}_{FE}$. If the problem is well formulated and the discretizations are properly selected then $\vec{u}_{FE} \to \vec{u}_{EX}$ as $N \to \infty$.

MSC/PROBE is based on the displacement formulation. For this reason in the following discussion only the displacement formulation is considered. In the displacement formulation the difference between $\vec{u}_{EX}$ and $\vec{u}_{FE}$ is naturally measured in the energy norm. By definition, the energy norm of a displacement function $\vec{u}$ is the square root of the strain energy of $\vec{u}$, which is usually denoted by $\|\vec{u}\|_E$. In the displacement formulation $\vec{u}_{FE} \to \vec{u}_{EX}$ in the following sense:

$$\lim_{N\to\infty} \|\vec{u}_{EX} - \vec{u}_{FE}\|_E = 0. \tag{1}$$

The choice of a particular discretization involves the creation of a set of functions S on the solution domain Ω. S is characterized by the mesh Δ, the mapping functions $\mathbf{Q}$, that is the functions which map standard finite elements onto the elements of the mesh, and the function spaces defined on the standard elements, called standard spaces. Most commonly the standard spaces are polynomial spaces characterized by the polynomial degree p. Hence $S = S(\Omega, \Delta, \mathbf{Q}, \mathbf{p})$. By definition, $\mathbf{Q} = \{\mathbf{Q}^{(1)}, \mathbf{Q}^{(2)}, \ldots, \mathbf{Q}^{M(\Delta)}\}$, $\mathbf{p} = \{p_1, p_2, \ldots, p_{M(\Delta)}\}$ where $M(\Delta)$ is the number of elements in the mesh. In MSC/PROBE typically uniform p, that is $p_i = p$ $(i = 1, 2, \ldots, M(\Delta))$ is used.

The subset of functions in S which satisfies the kinematic boundary conditions is denoted by $\tilde{S}$. The number of degrees of freedom N is the number of linearly independent functions in $\tilde{S}$. The finite element solution is that function from $\tilde{S}$ which minimizes the error in energy norm:

$$\|\vec{u}_{EX} - \vec{u}_{FE}\|_E = \min_{\vec{u}\in\tilde{S}} \|\vec{u}_{EX} - \vec{u}\|_E. \tag{2}$$

The error $\|\vec{u}_{EX} - \vec{u}_{FE}\|_E$ can be reduced in various ways: by mesh refinement; changing the mapping; changing the standard spaces; increasing the degree of the standard polynomial space, or any combination of these. For reasons of implementation most commonly the mesh is refined, or the degree of the standard polynomial space is increased, or mesh refinement is combined with increase of the polynomial degrees. These approaches are respectively called *h-extension*, *p-extension* and *hp-extension*. When aspects of implementation rather than reduction of error are discussed then the word *version* instead of *extension* is used. From the point of view of implementation there are substantial differences between the h-version on one hand and the p- and hp-versions on the other.

THE DEFINITION OF MATHEMATICAL MODELS

Mathematical models are essentially transformations. They transform one set of data, the input data, into another set, the output data. Mathematical models cannot improve on the quality of the information which resides in the input data. They can, however, damage the input information so that the output information is useless and worse, misleading.

For models based on the displacement formulation it is necessary that the potential energy of the exact solution $\Pi(\vec{u}_{EX})$ be bounded from below and the energy norm measure of the exact solution $\|\vec{u}_{EX}\|_E$ be nonzero. In addition, the model has to be consistent with the goals of computation: If the goal is to compute certain functionals, such as, for example, stress maxima, then the functionals of interest computed from the exact solution must be finite.

The problem of proper model definition has received very little attention in the finite element literature. Often model definitions are based on intuitively plausible reasoning which violates one or more of the conditions listed in the preceding paragraph. The most common modeling errors are:

1. Use of concentrated forces in two- and three-dimensional elasticity and in plate/shell models based on the Reissner-Mindlin or higher theories. This violates the condition that the potential energy must be bounded from below.
2. Use of point constaints. Point constraints should be used only as rigid body constraints, that is the external loads must satisfy equilibrium. If this condition is not satisfied then, depending on other boundary conditions, either $\Pi(\vec{u}_{EX}) = -\infty$, or $\|\vec{u}_{EX}\|_E = 0$, or $\vec{u}_{EX}$ is not distinguishable in the energy space from the solution which would be obtained if the point constraint were not applied.
3. One or more functionals of interest computed from the exact solution are infinite. For example, the goal is to determine stress maxima in elasticity and the exact value of the maximum stress is infinity.

The finite element solutions and/or the functionals of interest computed from the finite element solutions based on such models are entirely discretization-dependent. The data computed from the finite element solutions can be of credible magnitude and therefore very misleading. Some specific examples are presented in [1,2]. One important advantage in using p-extensions is that conceptual errors in modelling and many types of input errors become quite visible when the convergence tests made possible by p-extensions are performed. Such errors are often overlooked when only one solution is available.

Given that h-, p- and hp-extensions are alternative approaches to controlling the errors of discretization, the question naturally arises: When and why should one choose h-extension, p-extension or hp-extension? This question is now examined from the theoretical and practical points of view. The essential difference between the theoretical and practical points of view is that theoretical estimates are concerned with the asymptotic behavior of the error measured in precisely defined norms, usually the natural norm of the formulation, whereas in practical analyses one is concerned with approximating certain functionals, typically to within one to five percent relative error. The functionals of interest may or may not be directly related to the natural norm, however they can be related to the natural norm by means of extraction procedures, an example of which is presented in this paper. Satisfactory approximations often can be achieved well before the asymptotic range is entered. Experience is an important a priori information for designing the discretization.

Classification of problems.

It is useful to classify the exact solution $\vec{u}_{EX}$ in relation to the finite element mesh into three categories:

Category A: $\vec{u}_{EX}$ is analytic† on each finite element, including the boundaries of each finite element.

Category B: $\vec{u}_{EX}$ is analytic on each finite element, including the boundaries of each finite element, with the exception of some of the vertices (in three dimensions also along some of the edges). The points where $\vec{u}_{EX}$ is not analytic are called *singular points.*

For problems in category B the exact solution in the neighborhood of singular points can be typically written as the sum of some smooth function and a function in the following form:

$$\vec{u}_{EX} = \sum_{i=1}^{\infty} A_i r^{\lambda_i} \vec{\Phi}_i(\theta) \qquad r > r_0 \tag{3a}$$

where r, θ are polar coordinates centered on the singular point; A_i are coefficients which depend on the loading; λ_i is a fractional number, greater than zero; $\vec{\Phi}_i(\theta)$ is a smooth or piecewise smooth function, and r_0 is the radius of convergenece. It is possible to determine λ_i and $\vec{\Phi}_i$ from the definition of the model. Details are discussed, for example, in [3]. Define:

$$\lambda = \min_i \lambda_i \qquad i = 1, 2, \ldots \tag{3b}$$

The smoothness of the solution in the neighborhood of the singular point is characterized by λ.

A problem is said to be *strongly in category B* if $\lambda < 1$ otherwise it is *weakly in category B.* This choice of subdivision of category B is made with reference to the fact that in computational mechanics one of the usual goals is to compute stress data which are related to the first derivatives of the displacement. For problems strongly in category B the maximum stress is infinity in the singular

† *A function is analytic in a point if it can be expanded into a Taylor series about that point.*

point, whereas for problems weakly in category B the stress is finite on the entire domain. If the goals were to compute, say, the second derivatives of the stress, then it would be logical to regard problems for which $\lambda < 3$ as being strongly in category B. Whether a problem is strongly or weakly in category B often depends on decisions concerning idealization. Given alternative choices of idealization, it is generally preferable to select the idealization so that λ is as large as possible [4].

Category C: The mesh cannot be constructed so that the points where the solution is not analytic are at vertices (or, in three dimensions, along element edges) or the locations where abrupt changes occur in the derivatives of $\vec{u}_{EX}$, such as material interfaces, are at interelement boundaries.

A problem is *strongly in category C* if the distribution of points where the solution is not analytic lacks any a priori recognizable pattern. A problem is *weakly in category C* if the points where the solution is not analytic are distributed in some regular pattern.

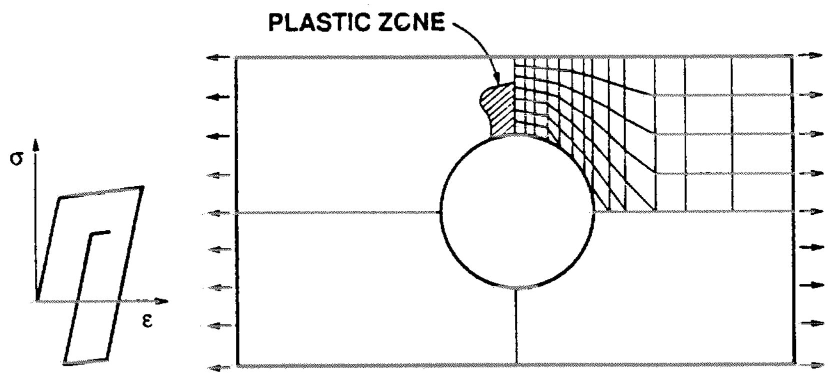

Fig. 1. Tension strip with a circular hole.

Consider, for example, the tension strip with a circular hole, shown in Fig. 1. Assume that the material is modelled by the elastic-strain hardening stress-strain law, shown in Fig. 1, and the material exhibits the Bauschinger effect. If the material is never loaded such that the yield point σ_{yield} is exceeded then the solution is very smooth and the problem is very weakly in category B. The reason that it is not in category A is that in points A and B the solution is of the form (3) with $\lambda \approx 2.75$, see for example [3]. If the load is increased past the yield point and kept constant, or cycled with a constant amplitude, the problem is weakly in category C. The singular points are the points on the boundary between the elastic and plastic regions which is, of course, solution-dependent. If the load is cycled with variable magnitude so that the yield stress is repeatedly exceeded in tension and compression then the problem is strongly in category C: the distribution of the singular points is dependent on the load history. In this case no pattern is discernible a priori for the distribution of singular points, except that there may be regions where the stress does not exceed the yield point, and the solution on those regions belongs in categories A or B.

A priori estimates of error.

Based on the classifications by categories A, B, C, close a priori estimates are available for the error measured in energy norm. These are *asymptotic* estimates, that is the estimates are close when N is sufficiently large. A priori estimates indicate the rate of change of the error with respect to increasing number of degrees of freedom. Clearly, it would be better to know the rate of change of the error with respect to some work measure, such as the number of operations, rather than the number of degrees of freedom, but such work measures are strongly implementation- and machine-dependent, hence difficult to interpret.

Using the asymptotic rate of change of the error measured in energy norm with respect to the number of degrees of freedom as the basis for comparison, the available a priori estimates provide a basis for the choice of extension process:

For problems in category A the most effective method for controlling the errors of approximation is by p-extension because the error, measured in energy norm, decreases *exponentially* when p-extension is used:

$$\|\vec{u}_{EX} - \vec{u}_{FE}\|_E \leq \frac{k}{\exp(\gamma N^{\theta})} \tag{4}$$

where k, γ, θ are positive numbers. If h-extension is used for problems in category A then the asymptotic rate of convergence is algebraic:

$$\|\vec{u}_{EX} - \vec{u}_{FE}\|_E \leq \frac{k}{N^{\beta}} \tag{5}$$

where k and β are positive numbers with $\beta = \min(p, \lambda)/2$.

For problems in category B the most effective method for controlling the errors of approximation is by hp-extensions: The mesh is graded so that the sizes of elements decrease toward the singular points in geometric progression with a common factor of about 0.15. Such meshes are called *geometric meshes.* The polynomial degree of elements is distributed linearly, with rounding to the nearest integer, such that the lowest polynomial degree is assigned to the smallest elements, the highest polynomial degree to the largest. If h- or p-extensions are used for problems in category B then the asymptotic rate of convergence is algebraic. If the singular points are nodal points then the asymptotic rate of convergence of p-extensions is characterized by $\beta = \lambda$, otherwise $\beta = \lambda/2$ in eq. (5). If the meshes are adaptively constructed for h-extensions then the asymptotic rate of convergence of h-extensions is characterized by $\beta = p/2$ otherwise by $\beta = \min(p, \lambda)/2$.

For problems strongly in category C h-extension is the best approach. In this case convergence is *algebraic*. For problems weakly in category C some combination of the h- and p-extensions or h- and hp-extensions is the best approach. In this case mesh refinement is used in those regions which contain the points where the solution is not analytic and p- or hp-extension is used elsewhere. The asymptotic rate of convergence is algebraic but much faster rate of convergence can be realized in the preasymptotic range [5].

Practical aspects.

Most problems in linear elastostatics and linear elastodynamics and many nonlinear problems belong in category B and engineering accuracy can be achieved by p-extensions in the *preasymptotic range* if properly designed meshes are used. Hence the use of p-extensions and properly refined meshes are of substantial importance in engineering design and analysis. In fact, h-extension is not a good choice for such problems.

Particular choices of mesh depend on (1) what data are of interest; (2) what accuracy is desired, and (3) how the data are to be computed. For many problems in category B geometric refinement at singular points is not necessary and in many important cases even the requirement that a singular point has to be a mesh point can be relaxed: Engineering accuracy can be achieved with p-extension, and coarse finite element meshes.

Whether sufficient accuracy can be achieved by p-extension for a particular choice of mesh cannot be known a priori. Knowing the proper classification of the solution, and the proper meshing for that class, is important in guiding the refinement process. In general, it is best to start with a simple mesh and perform p-extension. The error in energy norm is estimated by the method described in the following section and convergence of the quantities of interest is observed. If convergence of the quantities of interest is realized, and the error in energy norm is small, then the discretization error for the mathematical model is small. If convergence is not realized, and/or the error in energy norm is large, then the mesh has to be refined. For best results mesh refinement should be patterned after the optimal meshing for hp-extensions: The mesh should be graded in geometric progression toward the singular points with a common factor of about 0.15. p-Extension is then performed on this fixed geometric mesh. The results are checked for accuracy and, if necessary, additional layers of elements, graded in geometric progression, are introduced at the singular points. The number of layers needed thepend on the coefficient of the singular term. In most cases two layers are sufficient. Once again, a sequence of solutions is obtained by p-extension. The strain energy of the error decreases exponentially, provided that there is a sufficient number of elements at the singular points.

THE ERROR ESTIMATION CAPABILITY OF MSC/PROBE

The error estimation capability of MSC/PROBE is based on the estimate (5) and the fact that p-extensions naturally create solutions corresponding to a sequence of spaces $S_1 \subset S_2 \subset S_3 \ldots$. It can be shown that

$$\| u_{EX} - u_{FE} \|^2_{E(\Omega)} = \Pi_p - \Pi \tag{6}$$

where Π_p is the potential energy computed for polynomial degree p and $\Pi \overset{\text{def}}{=} \Pi(\vec{u}_{EX})$ is the potential energy corresponding to the exact solution [6]. If p-extension is used for problems in category B then the asymptotic rate of convergence is algebraic, that is

$$\Pi_p - \Pi \le \frac{k^2}{N_p^{2\beta}}. \tag{7}$$

For sufficiently large N the 'less than or equal' sign can be replaced with 'approximately equal'. This is because (7) is a tight estimate for sufficiently large N. Using (7) for three finite element solutions corresponding to the spaces S_{p-2}, S_{p-1}, S_p, the constants k and β can be eliminated and the following relationship obtained:

$$\frac{\Pi - \Pi_p}{\Pi - \Pi_{p-1}} \approx \left(\frac{\Pi - \Pi_{p-1}}{\Pi - \Pi_{p-2}}\right)^Q \tag{8}$$

where Q depends only on N_{p-2}, N_{p-1}, N_p:

$$Q \overset{\text{def}}{=} \frac{\log(N_{p-1}/N_p)}{\log(N_{p-2}/N_{p-1})}. \tag{9}$$

To obtain an estimate of the exact potential energy Π, the nonlinear equation (8) has to be solved. Although this estimator is asymptotically correct only when p-extension is used for problems in category B, computational experience has shown it to be reliable and generally accurate in the preasymptotic range and for problems in category A as well. For problems in category A the estimator tends to overestimate the error by a small margin. For problems in category B the estimator slightly overestimates the error in the preasymptotic range. It is highly accurate in the asymptotic range, but underestimates the error, by a small margin, in the the region of transition from the preasymptotic to the asymptotic range [6,7].

EXAMPLE: ATTACHMENT LUG

An attachment lug, typical of problems weakly in category B, is shown in Fig. 2a and the corresponding mesh in Fig. 2b. The lug is of uniform thickness of 0.5 in. Plane stress conditions are assumed. The modulus of elasticity is 3.0×10^4 k/in^2 and Poisson's ratio is 0.3. The goal is to find the magnitude and location of the largest tensile stress in the neighborhood of the circular hole when a sinusoidal normal pressure is applied on the inside perimeter of the hole, as shown in Fig. 2a. The pressure distribution $T_n = T_n(\theta)$ (k/in^2 units) is given by the expression:

$$T_n = \begin{cases} -\dfrac{32}{\pi}\cos\theta & -\dfrac{\pi}{2} \le \theta \le \dfrac{\pi}{2} \\ 0 & \text{otherwise.} \end{cases} \tag{10}$$

The pressure distribution may vary such that the direction of its resultant ($F =$ 10 k), characterized by the angle α, is in the range of ± 45.0 degrees.

Generally a lug is part of a larger structural unit, such as a bulkhead in an airframe. It would be impractical to model the entire structural unit just to find the stress distribution in the neighborhood of the circular hole of the lug. For this reason some boundary condition is imposed on the lug along the interface between the lug and the structure to which it is attached. The choice of this boundary condition is a modelling decision which, together with the other input data, determines $\vec{u}_{EX}$. The underlying assumption is that the data of interest are not sensitive to the modelling decision. It is necessary to check whether this assumption is valid: A mathematical model cannot be considered as reliable if the data of interest are sensitive to arbitrary modelling decisions.

Two modelling decisions are compared in the following: First it is assumed that the lug is held in equilibrium by smoothly varying tractions imposed along the boundary AB. In this case the exact solution is weakly in category B. Second, it is assumed that the lug is fixed along AB. In this case the exact solution is strongly in category B. In reality, the lug is elastically constrained so that smaller deformation is allowed along AB than in the first case but larger than in the second.

Equilibrium loading.

The lug is held in equilibrium by linearly distributed normal tractions and quadratically distributed shear tractions applied along AB:

$$T_x = -\frac{10}{3}\cos\alpha + \frac{25}{3}\sin\alpha(y-3) \tag{11a}$$

$$T_y = -\frac{5}{9}y(6-y)\sin\alpha. \tag{11b}$$

These tractions are consistent with the simple stress distributions of the usual engineering theory of beams. Only rigid body constraints are imposed.

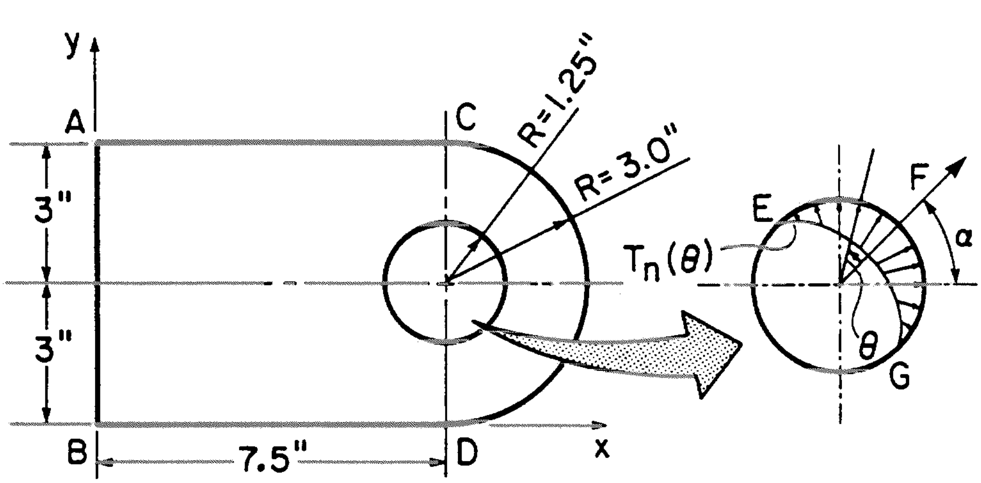

Fig. 2a. Attachment lug. Typical for problems weakly in category B.

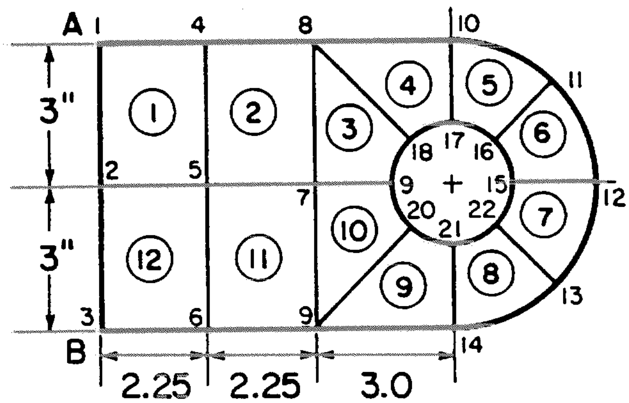

Fig. 2b. Finite element mesh for the attachment lug.

The problem has several weakly singular points. These are points A, B, C, D, E, G, shown in Fig. 2a. The location of points E and G depends on the choice of α. To realize the faster asymptotic rates of convergence of p- and hp-extensions, singular points should be nodal points. It would be inconvenient, even impractical, to have a different mesh for each α, however. It is now demonstrated through a computational experiment that if the goal of computation is to determine the location and magnitude of the largest tensile stress to within one percent relative error then points E and G do not have to be nodal points. This loading has substantial practical importance because it is typical for loadings of pinned structural connections.

First, p-extension is performed on the mesh shown in Fig. 2b and the relative error in energy norm is estimated using the procedure described in the preceding section. The relevance of this estimate for this particular problem is that the error in energy norm is closely related to the root-mean-square error in stresses, hence it is indicative of the average error in stresses [7]. The computations were performed with MSC/PROBE. The results, shown in Table 1, indicate that the relative error in energy norm is under one percent at $p = 8$.

Second, the location and magnitude of the largest principal stress is computed and p-convergence is checked. These computations involve the use of a data

Table 1. Attachment lug. Equilibrium loading, $\alpha = \pi/6$.
Estimated relative error in energy norm.

p	N	$\|\vec{u}_{FE}\|_E^2$ (in k)	Rel. Error (percent)
1	41	1.46368×10^{-2}	50.29
2	109	1.89925×10^{-2}	17.47
3	177	1.94095×10^{-2}	9.61
4	269	1.95218×10^{-2}	5.92
5	385	1.95680×10^{-2}	3.39
6	525	1.95857×10^{-2}	1.57
7	689	1.95891×10^{-2}	0.87
8	877	1.95900×10^{-2}	0.51

mesh, constructed as follows: Each element in the mesh is related to the standard quadrilateral element by the mapping:

$$x = Q_x^{(k)}(\xi, \eta) \qquad y = Q_y^{(k)}(\xi, \eta) \qquad -1 \le \xi, \eta \le +1 \tag{12}$$

where k is the element number. Each of the intervals $-1 \le \xi \le +1$, $-1 \le \eta \le +1$ are subdivided into n subintervals, thereby creating an $n \times n$ data mesh on the standard quadrilateral element. The stresses are then computed in each nodal point of this data mesh for each element and the location and magnitude of the largest principal stress is identified with respect to the data mesh. Of course, the resolution depends on n and the size of the elements. For this example $n = 10$ was selected.

Table 2. Attachment lug. Equilibrium loading, $\alpha = \pi/6$.
Location and magnitude of the largest tensile stress (σ_1)

p	Element number	x (in)	y (in)	σ_1 (k/in^2)	σ_2 (k/in^2)	angle (degrees)
1	4	6.616	3.884	19.64	3.87	38.1
2	8	8.067	1.886	19.76	2.83	27.7
3	8	8.067	1.886	22.28	3.59	27.9
4	8	8.067	1.886	22.81	3.51	27.2
5	8	8.153	1.934	22.29	0.95	31.1
6	8	8.153	1.934	22.34	0.33	31.1
7	8	8.067	1.886	22.32	0.33	27.1
8	8	8.153	1.934	22.32	−0.21	31.3

The results, shown in Table 2, indicate that the location of the largest principal stress converges to a point which lies in the vicinity of points (8.067, 1.886) and (8.153, 1.934). Both points lie on the perimeter of the circular hole. Since the data mesh is characterized by $n = 10$, the angular resolution along the perimeter of the circular hole is 4.5 degrees (0.098 in). These points are very close to $\theta = -\pi/2$ from the line of action of the applied force which is shown in Fig. 2a. It is seen that the location and magnitude of the maximum principal stress are virtually independent of the discretization: They change very little with respect to increasing p. The stress σ_2 is the minor principal stress. The angle between the direction of the largest principal stress and the x-axis is in the last column in Table 2. From these results it is possible to conclude that the largest principal stress occurs at the perimeter of the circular hole, its angular position is approximately $\theta = -\pi/2$ from the line of action of the applied force and its magnitude is approximately 22.3 k/in^2. Convergence is very fast: Good engineering accuracy is realized at $p = 3$. However, to verify that convergence has in fact occurred, extension must be continued for at least two p-levels beyond the p-level at which the desired accuracy has been reached. In this case p-extension could have been stopped at p=5.

On comparing the results in Tables 1 and 2 it is seen that the error in the maximum tensile stress is much smaller than the error in energy norm. The error in energy norm is related to the root-mean-square error in stresses over the entire domain. Locally the stress can be less accurate or, as in this case, more accurate.

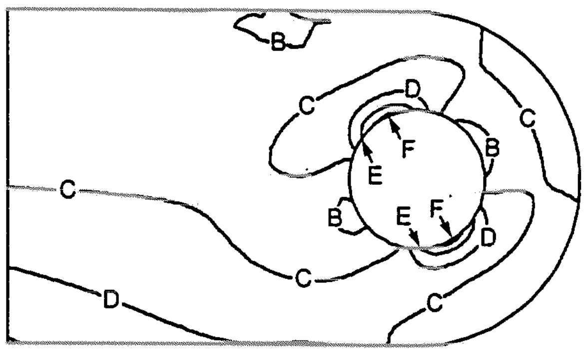

Fig. 3. Attachment lug. Contours of the first principal stress, $\alpha = \pi/6$. Contour interval: 5.0 k/in^2, $B = 0$, $F = 20.0$ k/in^2.

A contour plot of the largest principal stress, generated from stress values computed directly form the finite element sc'ıtion in the nodes of a 10×10 data mesh per element, is shown in Fig. 3.

Zero displacement along AB.

In this case the problem is strongly in category B. The computed values of the strain energy and the estimated relative error in energy norm are shown in Table 3. It is seen that convergence is somewhat slower, nevertheless reasonably good accuracy is achieved in energy norm.

Search for the location and magnitude of the maximum principal stress, restricted to elements 3 to 10, indicated strong convergence and yielded virtually the same results as those listed in Table 2. Hence the computed data are insensitive to both the modelling assumptions and the discretization.

Cracked attachment lug.

Assume now that a crack, 0.5 inches long, has developed in the lug, as shown in Fig. 4. The same kind of sinusoidal loading is applied as before, however in this case $\alpha = \pi/4$. The goal of computation is to determine the mode I and mode II stress intensity factors, respectively denoted by K_I, K_{II}. Two extraction methods, called cutoff function method (CFM) and the contour integral method (CIM), were used, detailed description of which is given in [8]. The mesh is now modified so that it is

Table 3. Attachment lug. Fixed along AB, $\alpha = \pi/6$.
Estimated relative error in energy norm.

p	N	$\|\vec{u}_{FE}\|_E^2$ (in k)	Rel. Error (percent)
1	38	1.44166×10^{-2}	50.64
2	102	1.86780×10^{-2}	19.15
3	166	1.91132×10^{-2}	11.93
4	254	1.92667×10^{-2}	7.94
5	366	1.93373×10^{-2}	5.16
6	502	1.93673×10^{-2}	3.35
7	662	1.93773×10^{-2}	2.47
8	846	1.93822×10^{-2}	1.88

geometrically graded at the crack tip, which is typical for hp-extensions, however p-extension is used on this mesh. This mesh is comprised of forty elements.

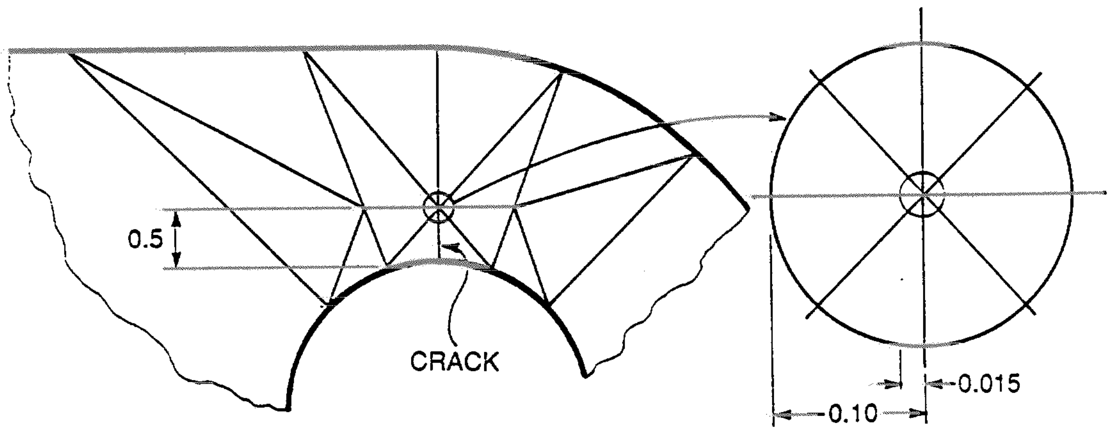

Fig. 4. Cracked attachment lug. Mesh detail.

The results given in Table 4 show that the stress intensity factors computed by the cutoff function method (CIM) and the contour integral method (CFM) methods converge strongly and obviously, although not monotonically. Greater accuracy and more nearly monotonic convergence is exhibited by the cutoff function method than the contour integral method. Both methods yield solutions which are within the range of precision normally needed in engineering computations at $p = 4$. The data for $p > 4$ merely confirm that convergence has in fact occurred.

NOTES ON IMPLEMENTATION

From the point of view of implementation there are several important differences between the h- and p-versions. One of these is the difference in mapping requirements: In the p-version the size of the elements is not reduced as the degrees of freedom are increased, hence the geometric description must be independent of the number of elements. For this reason mapping by the blending function method is used. In the case of the lug problem the circular boundaries were represented exactly in the computation of the stiffness matrices and load vectors.

Table 4. Cracked attachment lug, 40 finite elements.
p-Convergence of stress intensity factors in ksi$\sqrt{\text{in}}$ units.

p	N	$K_I/\sqrt{2\pi}$ (CIM)	$K_I/\sqrt{2\pi}$ (CFM)	$K_{II}/\sqrt{2\pi}$ (CIM)	$K_{II}/\sqrt{2\pi}$ (CFM)
1	93	5.335	4.252	−2.010	−1.607
2	269	3.492	3.851	−1.715	−1.866
3	473	3.954	3.836	−1.924	−1.842
4	757	3.731	3.785	−1.818	−1.845
5	1121	3.802	3.779	−1.852	−1.840
6	1565	3.773	3.783	−1.836	−1.841
7	2089	3.789	3.785	−1.845	−1.842
8	2693	3.783	3.785	−1.842	−1.843

For the same reason, representation of the loading must be independent of the number of elements. In the example of the attachment lug, the loading was specified by the formulae (10) and (11a,b). Terms of the load vector were computed by Gaussian quadrature, using 12 quadrature points for each loaded element side. When an element is only partially loaded, as in the case of $\alpha = \pi/6$ elements 4 and 8 are, then the normal pressure is zero in some of the Gauss points. This causes some perturbation in the integrand. Nevertheless, as seen in this example, this affected neither the overall quality of the solution, measured in energy norm, nor the accuracy of the largest principal stress significantly. The formula is most conveniently defined in a local coordinate system, rotated by the angle α in relation to the global system. In this way the orientation of the applied load can be changed by changing only a single input parameter α.

In the h-version stresses are usually evaluated at Gauss points only. For other points the stresses are determined by some smooth interpolation. There is no advantage in doing this in the p-version. In this example, the stresses were evaluated on the data mesh directly. No smoothing or averaging was performed. In fact, at interelement boundaries the stresses were evaluated independently for each element. The smoothness of the contour lines in Fig. 3 indicates that there are no significant jumps in the computed stress values between adjacent elements, which is another indicator that the solution is of good quality.

COMPARISON WITH OTHER METHODS

The objection has been raised in some discussions on the relative performance of the h- and p-versions that in the p-version the element stiffness matrices are much larger and many more quadrature points are used, hence the computer times are longer. This is, indeed, the case if the comparisons are based on equal number of degrees of freedom. It is more realistic, however, to base comparisons on equal accuracy. In that case the comparison favors the p-version since the rate of increase of computer time with respect to p is algebraic, whereas the rate of decrease of the error is exponential. Such comparisons are also inadequate for measuring performance from the point of view of practical use of finite element methods, however.

Comparisons should be based on how much human time, machine time, and disk memory are required to obtain the engineering data of interest with reasonable certainty that the results are within the desired range of accuracy. The cost of running the lug problem, with the boundary along AB fixed, is detailed in the following and the reader is invited to make his or her own comparison. The author would welcome communication of results.

1. **Human time:** This is, by far, the most important item from the economical point of view, and also the hardest to quantify because the human time spent on any project of this nature is strongly skill-dependent. In this case the job was performed by a skilled user of MSC/PROBE (the author) and the entire time, including data preparation, actual execution, and post-solution tests took 22 minutes. In practical applications it is also very important to consider the time required to make modifications. In MSC/PROBE changing the orientation of the load (see Fig. 2a), or the size of the circle, involves changing one data item only. The same is true for changing the crack length in the case of the cracked attachment lug. Changing the location of the circle, to examine the effects of excentricity, for example, requires changing two data items.

2. **Machine time:** A VAXstation 3100 Model 40, VMS operating system was used. If a sequence of p values is specified then MSC/PROBE computes the stiffness matrix for the highest p only, and the rest of the stiffness matrices are obtained using the fact that the stiffness matrix corresponding to p-1 is embedded in the stiffness matrix corresponding to p. In this case a sequence of solutions ranging from p=1 to p=8 was obtained.

Generation of stiffness matrices:	79 CPU sec
Solution (p=1 to p=8):	55
All other operations:	29
Total:	163 CPU sec

3. **Disk space:**

Data base and reports:	570 blocks
Temporary (scratch) space:	2966
Total	3536 blocks

It is emphasized that this effort produced a *converging sequence of eight finite element solutions* from which the error was estimated and the accuracy of the data ascertained. The value of the information generated is much greater than if only one solution had been obtained.

CLOSING REMARKS

Mathematical models are reliable if the error measured in the natural norm of the formulation is small and the data of interest are insensitive to both the choice of discretization and the modelling assumptions. Assurance of reliability is a systematic process in which sensitivities to discretization and modelling assumptions are investigated. In practical engineering decision-making processes the elapsed time between a problem being stated and some decision having to be rendered is generally quite limited, hence investigation of sensitivities is feasible only if the model is efficient. For this reason reliability and efficiency are closely related in practical computations.

The p-version of the finite element method has a very important role to play in mechanical and structural analysis and design: The simplified input data generation and updating process and the rapidly converging sequence of solutions leading to error estimation and quality control procedures are the most important considerations in this regard.

REFERENCES

[1] Szabó, B. "On Errors of Idealization in Finite Element Analyses of Structural Connections", Proceedings, Workshop on Adaptive Methods for Partial Differential Equations, edited by J. E. Flaherty, P. J. Paslow, M. S. Shephard and J. D. Vasilakis, Society for Industrial and Applied Mathematics, Philadelphia pp. 15-28 (1989)

[2] Bortman, J. and Szabó, B. A., "Structural Analysis of Fastened Joints", Proc., 1989 US Air Force Structural Integrity Program Conference, San Antonio, TX, Dec. 5-7, (1989).

[3] Williams, M. L., "Stress Singularities Resulting from Various Boundary Conditions in Angular Corners of Plates in Extension" *Journal of Applied Mechanics*, Vol. 19, pp. 526-528 (1952).

[4] Szabó, B. A., "Geometric Idealizations in Finite Element Computations", *Communications in Applied Numerical Methods*, Vol. 4, pp. 393-400 (1988).

[5] Oden, J. T., "Progress in Adaptive Methods in Computational Fluid Dynamics", *Adaptive Methods for Partial Differential Equations*", J. E. Flaherty, M. S. Shephard and J. D. Vasilakis, editors, Society for Industrial and Applied Mathematics, Philadelphia pp. 206-252 (1989).

[6] Szabó, B. A. and Babuška, I., *Finite Element Analysis,* Manuscript of book to be published by John Wiley in 1990.

[7] Szabó, B. A., "Mesh Design for the p-Version of the Finite Element Method", Computer Methods in Applied Mechanics and Engineering, Vol. 55, pp. 181-197 (1986).

[8] Szabó, B. A. and Babuška, I., "Computation of the Amplitude of Stress Singular Terms for Cracks and Reentrant Corners" *Fracture Mechanics: Nineteenth Symposium, ASTM STP 969*, T. A. Cruse, Editor, American Society for Testing and Materials, Philadelphia, pp. 101-124 (1987).