EVALUATION OF THE P-VERSION

OF THE FINITE ELEMENT METHOD

by

KAMRAN IZADPANAH

MSC

EVALUATION OF THE P-VERSION OF THE FINITE ELEMENT METHOD

INTRODUCTION

The field of finite element analysis has been the subject of intense development for many years [1, 2] and this tool in conjunction with the powerful computers available today, make finite element analysis an important part of the engineering analysis field. The methodology that most of the current commercially available codes, such as MSC/NASTRAN, are based upon is the so called h-version methodology which is based upon discretization of an engineering problem into many small finite elements (h) and assuming a linear or at most quadratic distribution of the unknown parameters in those small finite elements. The success of this methodology has been so great that little attention was given to the p-version of the finite element method in which the distribution of the unknown parameters on the finite elements were no longer restricted to linear or quadratic, but to an arbitrary higher order p. This methodolgy, which was virtually abandoned in the early stages of the development, has made some very interesting advances in the last few years [3, 4, 5] and is the topic of this paper.

Briefly, the advances in the p-version methodology were demonstrated using high order elements with linear, quadratic, and in general, linear blending mapping techniques in 2-Dimensional elasticity problems [5]. The results of these studies indicated that:

1. Very good results can be obtained in smooth problems when the model is discretized by only the minimum number of quadrilaterals.
2. With a high enough p-order, the elements become less sensitive to shape.

3. Displacements and stresses can be computed accurately in the entire domain; in fact, relative errors of less than one percent were obtained for even singular problems.

Therefore, it was decided to perform a very complete evaluation of a p-version finite element program in order to understand its limitations and capabilities with the following goals:

1. Establish the numerical performance of the p-version FEM in broader and more general problems. The results of the numerical performance performed in this study are presented and examined in detail in Section 1.

2. Evaluate the architectural needs of such a code for possible incorporation in MSC/NASTRAN. These results are presented in Section 2.

3. Comment on the possibility of incorporating such a program in a small to medium size CAD/CAM system.

1.0 EVALUATION OF THE QUADHP ELEMENT

1.1 INTRODUCTION

In this section a series of numerical experiments are used to evaluate the high order element QuadHP, "Quadrilateral - Hierarchic - Pconvergence", obtained from a prototype FEM program. The experiments were designed to examine static-membrane, dynamic-membrane, static-plate bending and dynamic-plate bending (Reissner-Mindlin formulation) of the element and were performed on typical engineering problems with real applications.

The three problems shown in Figure 1 (described in detail in Section 1.2.1) were felt to be sufficient to demonstrate the capabilities and the deficiencies of the QuadHP element in static-membrane conditions and will be analyzed in detail. The static-plate bending behavior of the QuadHP was evaluated by analyses of the infamous Rhombic Plate problem (Figure 2). The Rhombic Plate problem (described in Section 1.2.1) has been used for this purpose for many years and an immense amount of literature is available on the performance of other elements [8]. A brief review of this literature showed that one of the best performing displacement-based finite elements is the MSC/NASTRAN QUAD4 element. Hence, comparison will be made between the QuadHP and QUAD4.

In the case of dynamic-membrane analyses, however, the selection of test problems is much more difficult. To the author's knowledge, there are no standard problems proposed in the literature that could be used for comparison. It was therefore decided that, a good measure of performance would be the first three eigenvalues of a simply supported plate (Figure 3). In this section of the evaluation, the eigenvalues of the QuadHP element will be compared with the QUAD4 element. The problem of selecting test problems for comparing dynamic-plate bending analyses is even more difficult because in addition to lack of standard test problems, it was discovered that in most programs, including MSC/NASTRAN, the inertia due to the rotational degrees of freedom are neglected and comparison of the results obtained from QuadHP with their results would be unfair. It is noted here, however, that lower eigenvalues, i.e., a better approximation, were obtained in plate bending problems with the QuadHP and compared to QUAD4 and QUAD8 elements in all problems investigated.

1.2 MEMBRANE ANALYSIS

1.2.1 Static Problems

The problems selected for the static-membrane analyses, as shown in Figure 1, are:

1. Finite width strip under tension with circular hole. This is a typical problem in aerospace structures. The stress concentration of this problem will be analyzed for a variety of r/w ratios.

2. Thin circular arch loaded at one end. This problem was recommended as a test problem for quadrilaterals [6]. The vertical displacement of the point under the load will be examined.

3. Thick walled cylinder under internal pressure. This problem was recommended to examine the effects of nearly incompressible material [6]. A plane strain condition is used to intensify the numerical difficulties.

4. Thin circular arch under internal pressure. This problem will test the membrane behavior of the QuadHP. The outward displacement of the tube will be compared.

The mesh design for Problem 1 is shown in Figure 4. The NASTRAN mesh consists of 334 elements and the QuadHP mesh consists of the minimum number of quadrilaterals that could represent the geometry, i.e., 3. The results of computation of stress concentration factor at Point A are shown in Figure 1a plotted against the exact solution shown in a solid line. It can be observed that with P = 8 the QuadHP results

1. Are more accurate than the MSC/NASTRAN results.

2. Are accurate in a large range of r/w of .05 to .90.

It is noted here that in order to model the r/w of .99 and .05, major mesh modifications were necessary for the NASTRAN analysis, and therefore, the analyses were not performed. This was not the case with QuadHP since the only input value changing was the actual radius, and the wider range of r/w could be analyzed easily.

The mesh design of Problem 2 for MSC/NASTRAN is shown in Figure 6. It consists of 6 QUAD4 elements. The mesh for QuadHP consists of only 1 element and is similar to Figure 1b. The relative error in the computed displacement under the load is shown in Figure 7, in addition to results obtained from QUAD8 and PAL2 [7]. It can be observed that

1. QuadHP results are more accurate than the other elements.
2. A smooth monotonic convergence is achieved by QuadHP. This makes it possible to extrapolate these results for an even more accurate value.
3. The strain energy of the QuadHP is also converging monotonically. This cannot be said in general for QUAD4 or QUAD8. Monotonic convergence is considered a desirable but not a necessary feature of a good element.

The results of Problem 3 are shown in Table 1 in which the ratio of the computed values to the exact values are listed for the Poisson ratio of 0.49, .499, and .4999. The result of Poisson ratio of 0.4999999 is shown only for $p = 8$. It is observed that in all cases, with $p = 5$ and above, the relative error is less than 10% and at $p = 8$, the relative error is practically zero. This is not the case for Poisson ratio of 0.4999999 in which the relative error is about 8% at $p = 8$.

The results of Problem 4 are shown in Figure 8 for a 1 element QuadHP mesh. It can be seen that errors of less than 1% are achieved for this problem. The results were not compared with QUAD4 or QUAD8.

1.3 DYNAMIC PROBLEMS

The dynamic-membrane analysis was performed on a simply supported square plate shown in Figure 1c. This problem was analyzed with the QuadHP and a uniform mesh of 1, 4, 16, 64, 256 QUAD4 elements. The first five eigenvalues were compared but the results of the first three are shown in Figures 9 through 11. It is observed that

1. Both elements have monotonically converging eigenvalues.
2. The eigenvalues of QuadHP elements are smaller than QUAD4's for the same DOF and therefore the QuadHP's are more accurate.
3. The rate of convergence of the QuadHP elements is about twice that of QUAD4.

1.4 PLATE BENDING ANALYSIS

The simply supported thin plate is shown in Figure 2 with the corner angle varying between 90 (square) and 30 (diamond) degrees. Since this variation of the angle will cause the nature of the problem to change from smooth to singular, the elements will be tested in both situations. The error in the displacement at the center of the plate and the error in the principal moments at this point will be compared. The exact results were obtained from [8] for this purpose. The results of QUAD4 are given for a 4 x 4, 8 x 8, and 14 x 14 mesh configurations and will be compared to a single element mesh and a 9 element mesh (Figure 11). It is noted here that it is not necessary to increase the DOFs of QuadHP by a uniform mesh refinement since the user will have the capability to put the needed DOFs where they are needed most, i.e., the singularity. It is also noted that the results of the QuadHP for polynomial orders of less than 4 are not valid since severe locking effects will distort the results. However, as mathematically proven, this effect disappears after P = 4 and the results will be valid for comparison. (In the QUAD4 element, a selective integration scheme is used to alleviate this problem.)

The results of the displacement W, the maximum principal moment (Ma), and the minimum principal moment (Mb) using a single QuadHP and the QUAD4 elements are shown in Figures 12 through 20 for THETA = 90, 60, 30. (The results of THETA = 40, 80 are also available.) It is observed that in all cases the QuadHP results of displacements and moments are converging. More specifically, the relative error in the displacement W using the QuadHP element with about 100 DOF is the same as that of the QUAD4 elements at 560 DOF. This is not true for THETA = 30 degrees where the QuadHP achieves 15% error at 100 DOF while QUAD4 achieves 8% error at about 200 DOF and 11% error at 600 DOF.

The relative error in the principal moments Ma(maximum) and Mb(minimum), using the QuadHP element, is about the same as that of the QUAD4 elements but at about 1/2 to 1/3 of the number of degrees of freedom. This is not the case with THETA = 30 degrees where the QUAD4 elements obtain the same accuracy at the same NDOF in Ma but the QUAD4 results exhibit some oscillation in the Mb computations. It is noted here that the values of the minimum principal moment are very small in case of THETA = 30 degrees and the comparison of relative errors could be misleading due to the smallness of the absolute error involved.

The results so far have shown that a single QuadHP element performs extremely well in case of smooth plate bending problems (90 to 60 degrees), but for THETA = 30 a mesh refinement is recommended and necessary. The results of the 9 element QuadHP mesh (Figure 21) for THETA = 30 degrees is shown in Figures 22 through 24. It is observed that the increase in the NDOF of this mesh improves the computed displacements and moments with respect to the single QuadHP and QUAD4 elements. This improvement is apparent in both the error values achieved and the converging trend of the errors.

2.0 ARCHITECTURAL REQUIREMENTS

There are some fundamental architectural differences between a correctly constructed p-version finite element program and a conventional h-version finite element program such as MSC/NASTRAN even when the h-version programs include Serendipity or Lagrange type (8 or 12 noded) elements. The differences could be attributed to two major concepts in a p-version analysis which are not present in the h-version analysis, namely:

1. In a p-version program, there are some degrees of freedom (DOF) associated with the geometric entities of lines, surfaces, and volumes (corresponding to an element), in addition to the traditional DOFs associated with grid points.
2. The concept of a one-to-one correspondance between grid locations and the value of unity of the associated shape functions is no longer adhered to.

The degrees of freedom in a QuadHP element for each displacement field will be either corner DOFs, side DOFs (surface DOFs in 3-Dimensional analysis), or internal (bubble) DOFs. The corner, side, and surface DOFs can have boundary conditions and loads assigned to them in a similar fashion to that of the grid points in a grid-oriented analysis. For example, a 2-Dimensional QuadHP element can have at most 94 DOFs (two displacements) at p = 8 with the following distribution: 8 corner DOFs, 56 side DOFs and 30 internal (bubble) DOFs. In case of a plate bending anaylsis (one displacement and two rotations), this distribution will be 12 corner, 84 side, and 45 internal DOFs. An h-version program will have two DOFs for each grid and 4 or 8 grids per element which adds up to 8 or 16 total degrees of freedom per element for the 2-Dimensional case and three degrees of freedom per grid for the plate bending problem for a total of 12 or 24 DOFs.

Since the individual degrees of freedom of the p-version program would not have physical significance to the user, modeling techniques for external loads and enforced displacements would have to be provided. In the approaching era of fully automatic modeling this does not appear to be a disadvantage for the p-version method.

The second major difference between the p-version analysis and the h-version analysis deals with the original assumptions of the finite element method. Therefore, it is necessary to review some of the related areas of the theory in order to understand the consequences of this new approach. In the finite element method the unknown variable, e.g., displacement field U in an element, is assumed to have the form

$$U = \sum N_i U_i \qquad (1)$$

where U_i are the "nodal" diplacements and N_i are the shape functions of the polynomial of order p. In a linear element (p = 1), which is the parent element of all elements, it is assumed that the displacement field has a linear distribution and the shape functions (N_i) are constructed so that they have the value of unity at the ith node of the element. In a conventional quadratic element (p = 2), there are four more midside nodes supplied and the additional shape functions are so constructed as to have the value of unity at these additional nodes. An entire family of higher order elements could be constructed with the same concept in mind and depending on the technique used to construct the elements, this would result in a Serendipity or Lagrangian family of elements. Note that this technique of constructing additional shape functions and thus elements, does not take into account the numerical conditioning of the stiffness matrices generated, and if infinite precision were possible, the quality of the finite element analysis would have not been affected. In practice however, because of the finite digit arithmetic, the conditioning of the stiffness matrix of the elements is quite important and experience has demonstrated that the Serendipity and Lagrange family of high order elements could have numerical problems at p-orders of as low as 4 or 5. The cause of this ill-conditioning could be directly traced to the requirement on the shape functions of having a value of unity at a given "node" of the element (the polynomial selected for the 12th shape function was forced to have a value of unity at the 12th node). It is noted here that there is no theoretical need for this constraint since as long as the displacement field is interpolated over the element by the relation of Equation 1, the theory of the finite element method is valid and the convergence and stability cirteria are not altered. An analogy to this behavior is the improved numerical performance of a simple curve-fitting program when an orthogonal base, such as Legendre or Chebychef polynominals, are used as oppose to a normal polynominal expansion. The theoretical accuracy of the curve fitting is only governed by the polynominal order used, but the numerical behavior using the orthogonal system is far superior.

In the QuadHP element, the shape functions were constructed so that they produce a stiffness matrix which is nearly orthogonal with respect to the Laplace operator on a square domain. The Laplace operator was selected for this purpose because the theory of similar operators [10] predicts a similar numerical behavior from the shape functions in all the operators of this class. This means that N_i will no longer have the value of unity at the goemetrical location conventionally associated with the ith node but some other values like 0.5 or 0.3. The result of this selection is that the stiffness matrices obtained by using these shape functions are very well conditioned for all the solution squences dealing with differential equations related to the Laplace operator. This good numerical behaviour was demonstrated in the Rhombic Plate problem of Section 1 in which the material properties were so selected as to bring out numerical ill-conditioning of the stiffness matrices as well as the quality of the final displacement and stress values.

This method of constructing the shape functions has one additional benefit; it is no longer necessary to require n physical nodes to be input for a p-order element since the concepts of nodes can now be replaced by the concept of degrees of freedom and generated internally. This means that after the minimal geometrical information is supplied for an element (the 2-D QuadHP needs 8 nodes), the element geometry can be constructed. Next, depending on the order of p (p = 1 to 8 for the QuadHP) specified by the user, the degrees of freedom are generated internally. After the initial analysis is performed, the user can perform another analysis with more degrees of freedom (in most cases a more accurate analysis) by simply inputting a higher p-order for the elements involved. No other additional input data such as loads, displacements or boundary conditions are needed since these parameters are generated internally. This extension process can continue until the user is satisfied or the maximum polynomial order is used (p = 8). At this point he simply changes the geometrical input once more and starts again.

At this point, it is important to briefly discuss the continuity enforced in the p-version method. The QuadHP elements are C_0 elements; this means that there is no enforcement of higher order derivatives in the direction normal to the sides of the element. However, the displacement field along the side of two neighboring elements is matched exactly, and as a result, all the derivatives along that side will match. This is exactly the case in the h-version method since the displacement along the side of two neighboring elements is given as a linear or quadratic function of the nodal values associated with that side and hence, the displacement along that side is exactly the same. This type of continuity is mathematically the one that will result in the best numerical results since the space containing the permissible solutions will be as large as possible*.

The effects of implementing these concepts in a general purpose program, can be divided into two categories. The first one can be labeled the input/output data management category and includes the additional card images that have to be processed and appropriately stored in the data base of the program. This information would include for example, the loads and boundary conditions on the sides of the element, the order (p) of the element and so on. It is not difficult to design a data base to accomodate these needs of a p-version program.

*The solution u(FE) of a finite element analysis must have a finite strain energy associated with it. A simple and not very mathematical way to understand the effects of enforcing higher continuities is to consider all the functions satisfying this condition as "balls" and the space containing all these solutions as a "bowl". As higher continuity is enforced, there are more "holes" made in this "bowl" and it will be containing less "balls". This means that there will be less functions available to be the u(FE) and therefore the solution in general deteriorate.

The second category deals with the element stiffness generation and solution phase of the program. Here, fewer but much larger and denser stiffness matrices are generated which demand larger storage areas and use more CPU time. For example, a 2-Dimensional QuadHP element with p = 8 (maximum order) has a stiffness matrix of 94 X 94 with a density of between 17 to 40 percent depending on the distortion of the element. This compares with a linear element of size 8 X 8 for the h-version program. Note that even though the individual element stiffness matrices are large and the assembly phase per element of the matrices are more CPU intensive than that of the h-version programs, the assembled global stiffness matrix is of a reasonable size and the total assembly CPU time is about the same. This is because the total degrees of freedom for a given problem is usually 1/3 of the h-version analysis (in Problem 1 of Section 1 for example, the size of the global stiffness matrix is about 250 X 250). The only difference will be the bandwith of the p-version matrix which is larger than the comparable h-version matrix. This brings us to the final phase of this section, i.e., the solution phase. It must be clear that because of the larger bandwidth of the p-version global stiffness matrix, a conventional bandwith solver will not be as efficient and the CPU time for the solution will be larger than the comparable h-version matrix.** This topic needs to be discussed in more detail when the possibility of parallel processors in conjunction with the p-version is discussed.

In the data recovery phase of the analysis, e.g., stress computations there are less elements but more data that has to be used for each element.

**This is one of the reasons that the Frontal Solution Method [11] has sometimes been used for the solution of these global matrices.

At the same time however, there is no need for extrapolation or other smoothing techniques for stresses on the boundaries since these values are quite accurate for $p > 4$ or $p > 5$ when directly computed from the solution. In general, this section of the program is very similar to a conventional program, however, it offers improved post-processing capability.

3.0 MODELING 3-DIMENSIONAL STRUCTURES

The modeling technique used for a 3-Dimensional analysis of a structure is either a full 3-Dimensional analysis using solid finite elements or an analysis using a combination of shell and membrane elements. The full 3-D analysis is a very expensive operation and it is usually a last step in an analysis/design cycle. The other technique is much more reasonable since in most analyses it is possible to represent the structure in a two dimensional curvilinear coordinate system and model it with a combination of shell and membrane elements without an unacceptable loss of accuracy. It is therefore necessary for a general purpose finite element program to have both type of elements in order to give the analyst the opportunity to select the right element for the type of analysis he is performing. In this section the applicability of the p-version methodology to the construction of solid (3-D) and shell elements is addressed briefly and some of the difficulties that will arise are discussed.

First, let us discuss the 3-Dimensional elements. It should be quite clear that constructing a p-version isoparametric three dimensional element is conceptually simple and very straight forward. In fact, the same techniques which are used to convert the h-version 2-Dimensional elements to an h-version hexahedra elements are extended to handle the larger number of shape functions associated with the p-version element. The only parts that need additional attention are the formulation and the necessary input data of the linear blending mapping techniques for three dimensional structures. This is the capability which allows the user to define sides as being circular, spherical, etc., and was quite useful in the 2-Dimensional analyses. However, the stiffness and load vector matrices generated from the 3-Dimensional elements are even larger than of the 2-Dimensional elements and it is necessary to address the "BEST" computing environment for this type of analysis.

The recommended environment for any 3-Dimensional analysis in general is usually the large mini-computers, such as the VAX 11/780 series or larger main-frame computers. But even the performance of this type of machine becomes unacceptable for very large problems making the super computers, such as the CRAY machines, the ideal machines for this class of problems. An analyst using a 3-Dimensional p-version analysis will encounter very similar problems as the problem size increases and even though he is able to get more accurate results with less degrees of freedom with the p-version program, he can easily overburden a main-frame machine if he is not careful with the total number of degrees of freedom he chooses to model the structure. This can happen because he is not forced to refine the mesh he has selected by inputting more grid point locations which is quite difficult in a 3-Dimensional model, but by simply increasing the p-level of the elements already involved. As a result, it seems that the best environment for medium to large 3-Dimensional problems is the super computer class which allows the user to perform very accurate analyses with reasonable turnaround time.

The development of p-version shell elements is not as straight forward as the 3-Dimensional solid elements since many more questions have to be resolved before a general shell element can be constructed. These problems have nothing to do with the p-version methodology and are also unresolved in the h-version analysis. They deal mainly with the shell theories available in modern elasticity and the construction of a finite element which can model the shell effects, such as the transverse shear effects, and still pass the general requirements of the finite element method such as the patch test. These problems are still under research and even though many solutions have been proposed, it is felt that they are not yet suitable for implementing into an "IDEAL" p-version shell element.

4.0 CONCLUSION

The QuadHP element was tested thoroughly in Section 1 of this study and all the results obtained indicated that this method could be a very effective and powerful tool in the hands of a trained user. The excellent numerical performance of the program in the test problems of Section 1 in conjunction with a better pre- and post-processing possibilities of the program makes this program a commercially viable alternative to the conventional h-version finite element program. There are two additional topics that were not discussed in details but should be kept in mind in regards to the p-version analysis. These are automatic mesh generation and error estimates. The p-version is extremely well suited for automatic mesh generation because of the very few established and complete guidelines that are now available for this purpose [12]. These same guidelines are also the very essence of an error estimation section of the program which could help (or warn) the user/analyst in performing an analysis that can not be accurate using the selected mesh. One area that the p-version with or without error estimators will be quite effective is the field of shape optimization. Here, the design model and the finite element model will be very similar and include very few high order elements. These elements will be able to give accurate results to the optimization program even when they are extremely distorted since they are not very sensitive to this parameter. Next, the data from the optimizer could be directly used to construct the next configuration that has to be analyzed and therefore, the iteration process can conclude very quickly. Note, that if an error estimator is operational, the user will be warned when the model is no longer useful for the analysis and he can, with very few modifications, change the model and continue with the process. This capability is currently being investigated in a joint project between MSC and UCLA.

The discussions in Section 2 however, showed that a p-version program is a more detailed program than a conventional finite element program and more logic has to be incorporated in the code in order to take full advantage of the power of the method. With the increasing demand on a complete and user-friendly engineering workstation, this should not be viewed as a disadvantage since the geometric entities needed in the p-version program are very similar to the ones in existing CAD software and it is not difficult to visualize a CAD system using a p-version program as its very accurate analysis system. This possibility becomes even more enticing when a multi-processor environment is considered for this purpose since it is quite obvious that the architecture of a p-version program is very suitable for a parallel processor environment. For example, it is feasible to assign a separate processor to each element of the problem which will expedite the generation of the stiffness matrices and possibly the solution of the global stiffness matrix through a disection method.

It is also possible to incorporate this technology in an existing finite element program if the architecture of the program is modified as to have the ability to handle the specific needs of the p-version program. In fact, successful but awkward runs have been made by declaring the DOFs obtained from the QuadHP element as Scalar Points in MSC/NASTRAN and in some other instances the eigenvalue solutions of Section 1 were checked using the eigenvalue solution capabilites of MSC/NASTRAN.

The following recommendations are made as to possible future developments for this product. First, the feasibility of a full incorporation into MSC/NASTRAN should be more thoroughly investigated. At the same time, the feasibility of a smaller program in conjunction with a CAD software should also be studied which should also include the implications of using a multi-processor system. It is felt that this technology will be effective in both environments with the MSC/NASTRAN version responsible for large full scale analysis (3-Dimensional) and the CAD system serving as a tool for small to medium size problems.

References

1. Cook, R. D., "Concepts and Application of the Finite Element Method", Wiley, 1974.

2. Zienkiewicz, O. C., "The Finite Element Method", MacGraw-Hill, 1979.

3. Babuska I., Szabo B. A., Katz I. N., "The p-Version of the Finite Element Method", SIAM J. Numer. Anal. 18, 1981.

4. Babuska, I. and Szabo, B. A., "On the Rates of Convergence of the Finite Element Method", Intern, Journal of Numerical Methods in Engr. 18, 1982.

5. Izadpanah, K., "Computation of Stress Components in the p-Version of the Finite Element Method", DSc. Dissertation , Washington University, St. Louis, 1984.

6. MacNeal, R. H. and Harder, R. L., "A Proposed Standard Set of Problems to Test Finite Element Accuracy", Finite Elements in Analysis and Design, Vol 1, 1985.

7. MSC/pal2 Software.

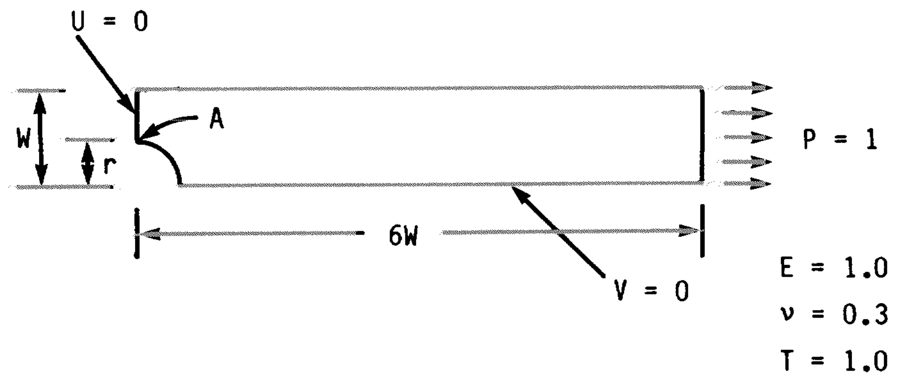
8. Morley, L. S. D., "Skew Plates and Structures", Pergamon Press, 1963.

9. Babuska, I. and Aziz, A. K., "On The Angle Condition in the Finite Element Analysis", SIAM J. Numer. Anal. 13, 1976.

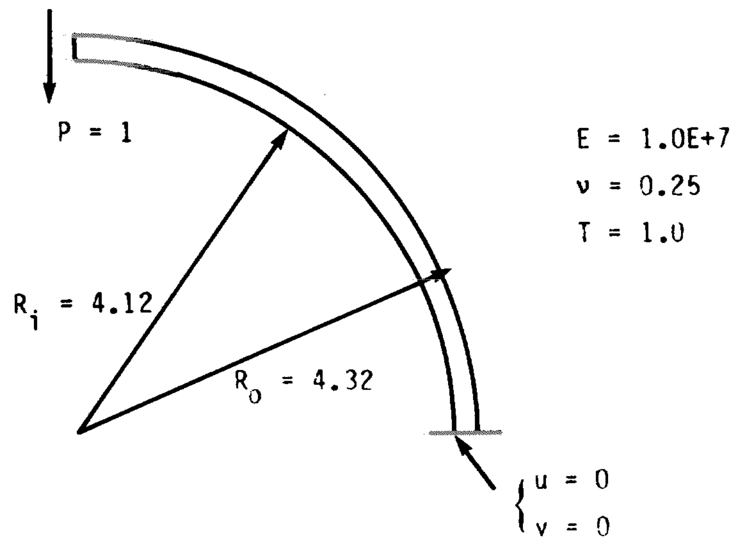
10. Mikhlin, S. G., "Variational Methods in Mathematical Physics", Pergomon Press, 1964.

11. Irons, B. M., "A Frontal Solution Program for Finite Element Analysis", International Journal for Numerical Methods in Engineering, Vol. 2, 1970.

12. Babuska, I. and Gui, W., "Basic Principles of Feedback and Adaptive Approaches in the Finite Element Method", Institute for Physical Science and Technology, Laboratory for Numerical Analysis, Tech. Note BN-1042, University of Maryland.



1a. Circular hole in a strip with finite width under tension.



1b. Circular arch loaded at free end.

Figure 1. Static Test Problems.

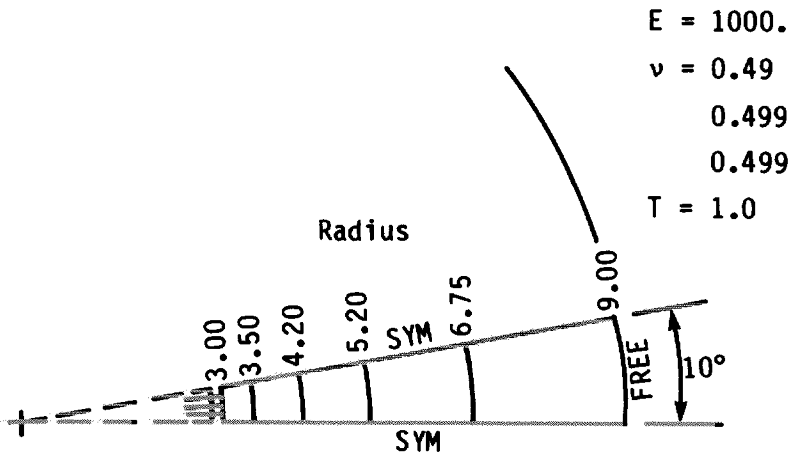
8. Morley, L. S. D., "Skew Plates and Structures", Pergamon Press, 1963.

9. Babuska, I. and Aziz, A. K., "On The Angle Condition in the Finite Element Analysis", SIAM J. Numer. Anal. 13, 1976.

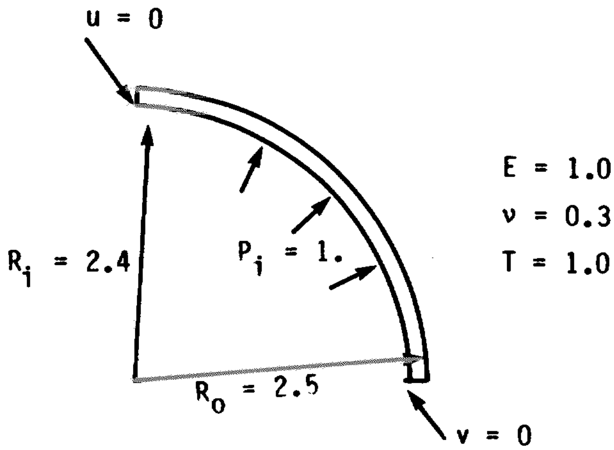
10. Mikhlin, S. G., "Variational Methods in Mathematical Physics", Pergomon Press, 1964.

11. Irons, B. M., "A Frontal Solution Program for Finite Element Analysis", International Journal for Numerical Methods in Engineering, Vol. 2, 1970.

12. Babuska, I. and Gui, W., "Basic Principles of Feedback and Adaptive Approaches in the Finite Element Method", Institute for Physical Science and Technology, Laboratory for Numerical Analysis, Tech. Note BN-1042, University of Maryland.



1c. 10° slice of a thick tube under internal pressure.



1d. Thin cylinder under internal pressure.

Figure 1. Static Test Problems (Cont.).

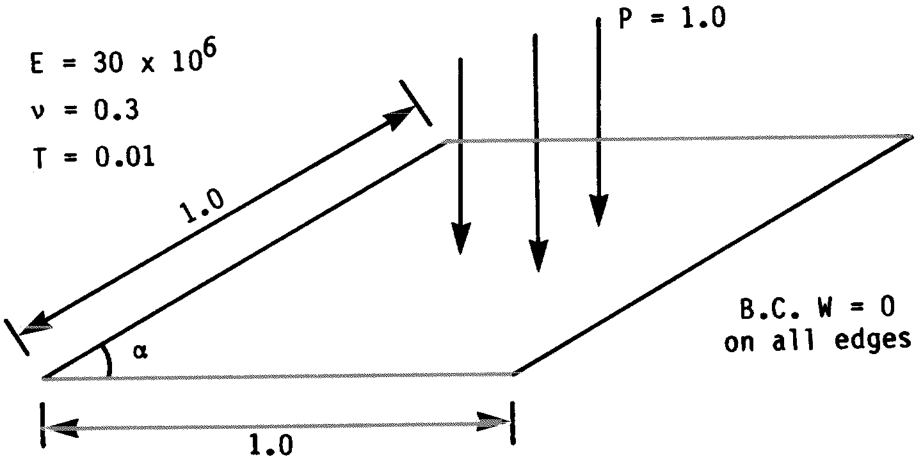


Figure 2. Rhombic Plate under Uniform Pressure.

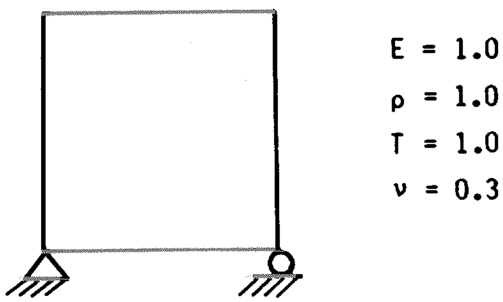
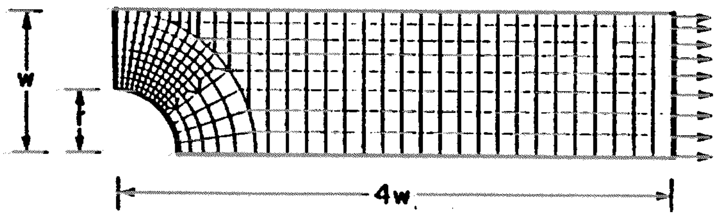
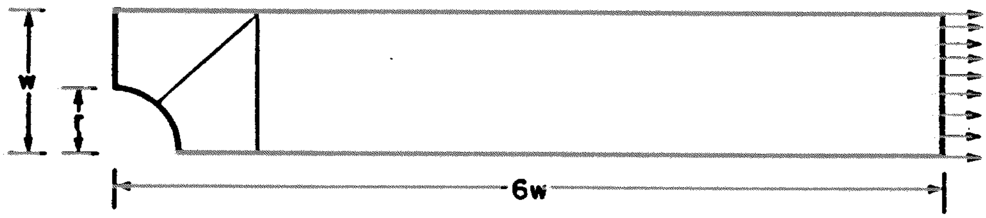


Figure 3. Eigenvalue Analysis Problem of Simply Supported Plate.



MSC/NASTRAN MESH
(334 Elements)



QUADHP MESH
(3 Elements)

Figure 4. Mesh Design for Problem 1a.

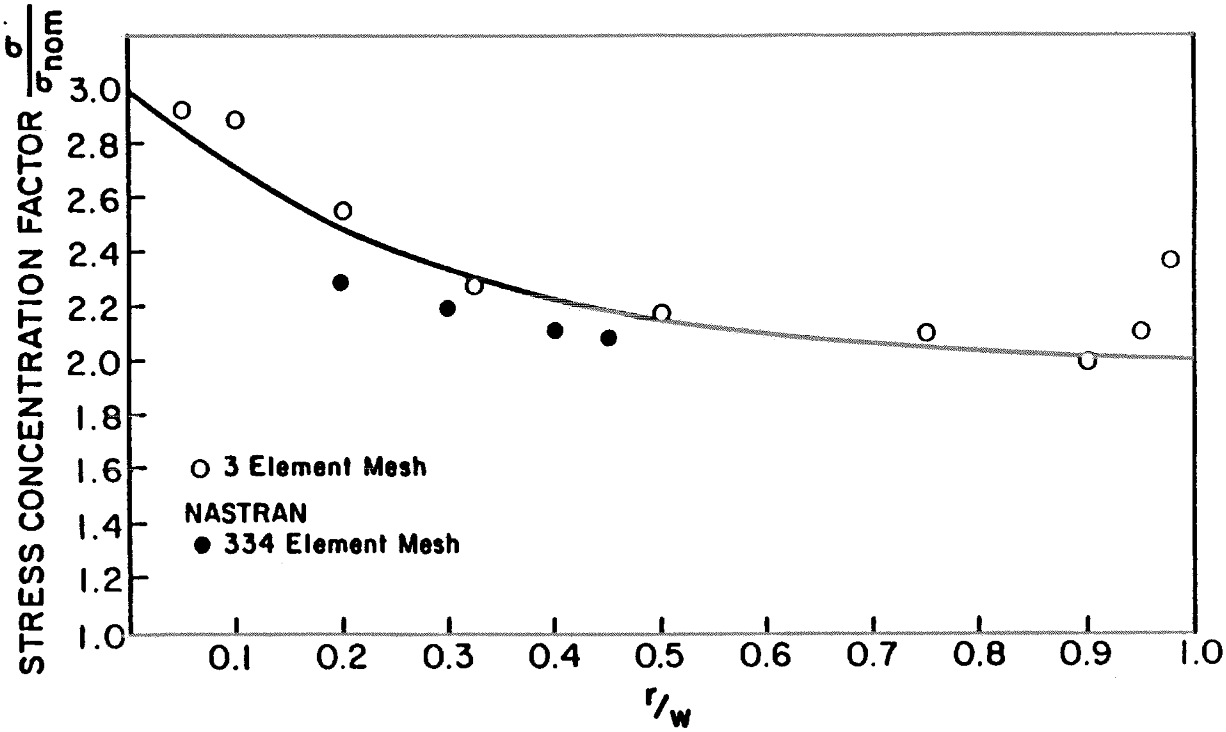
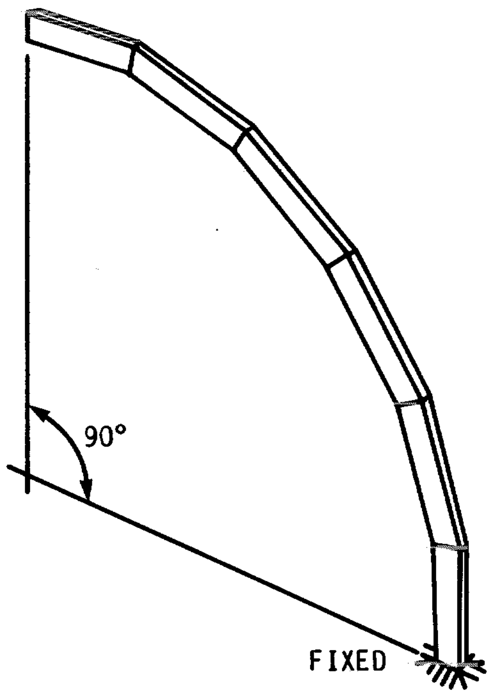


Figure 5. Stress Concentration Factor at Point A for Problem 1a.



Curved Beam

$R_i = 4.12$

$R_o = 4.32$

$T = 0.1$

$E = 1.0 \times 10^7$

$\upsilon = 0.25$

MSC/NASTRAN MESH = 6.1

Loading: unit forces at tip

Figure 6. MSC/NASTRAN Mesh Design for Problem 1b.

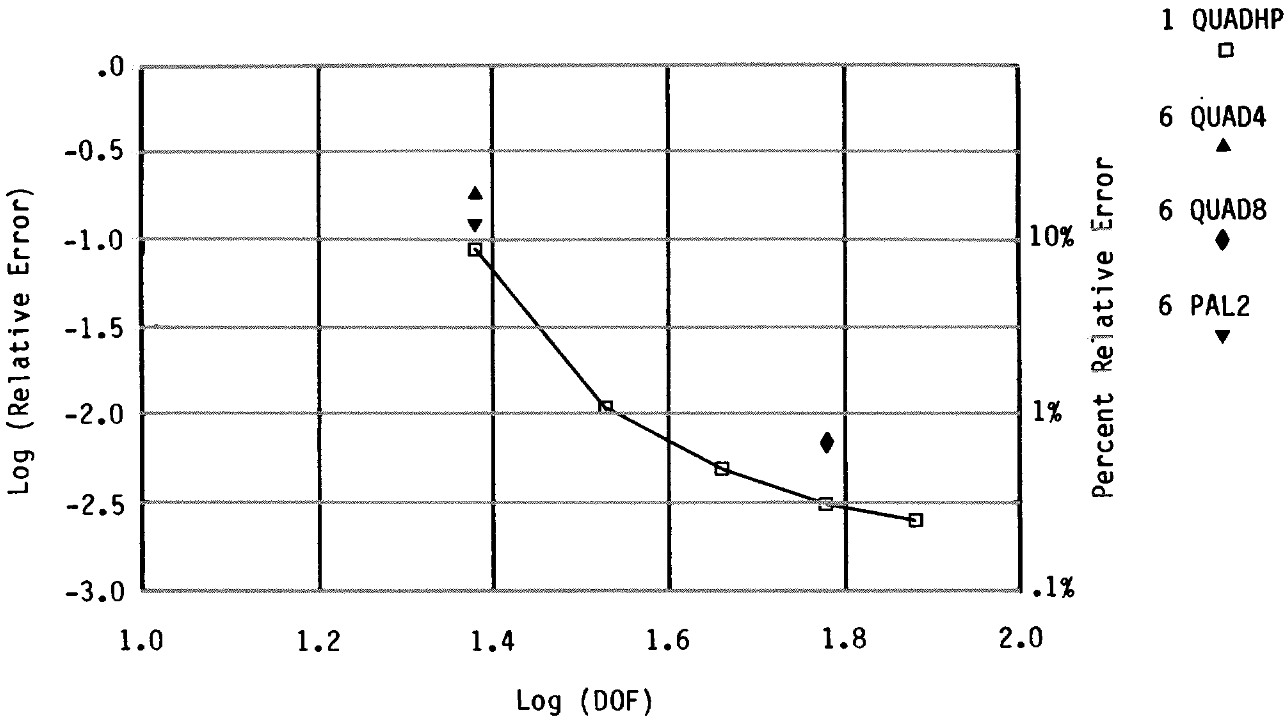


Figure 7. Relative Error of the Computed Displacement under the Load for Problem 1b (Circular Arch Loaded at End)

Table 1. Ratio of the Computed to Exact Displacement of Problem 1d.

		NU=0.49	0.499	0.4999	0.4999999
P	NDOF	U1/Uex	U1/Uex	U1/Uex	U1/Uex
1	2	0.168	0.020	0.002	
2	10	0.578	0.122	0.014	
3	16	0.916	0.523	0.099	
4	24	0.990	0.907	0.494	
5	34	0.999	0.990	0.904	
5Q8	34	1.000	0.997	0.967 ← NASTRAN WITH 5Q8	
6	46	1.00	0.999	0.990	
7	60	1.00	1.000	0.999	
8	76	1.00	1.000	1.000	0.918

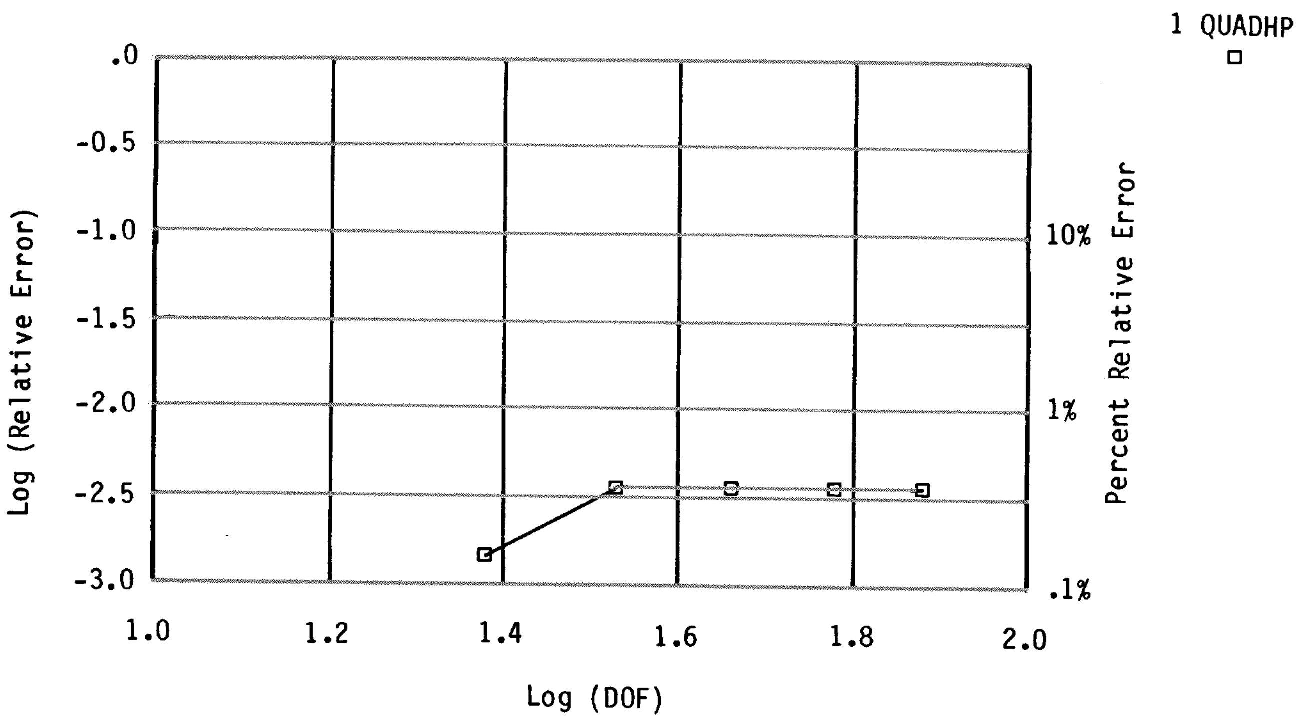


Figure 8. Relative Error of the Computed Radial Displacement of the Thin Cylinder under Internal Pressure.

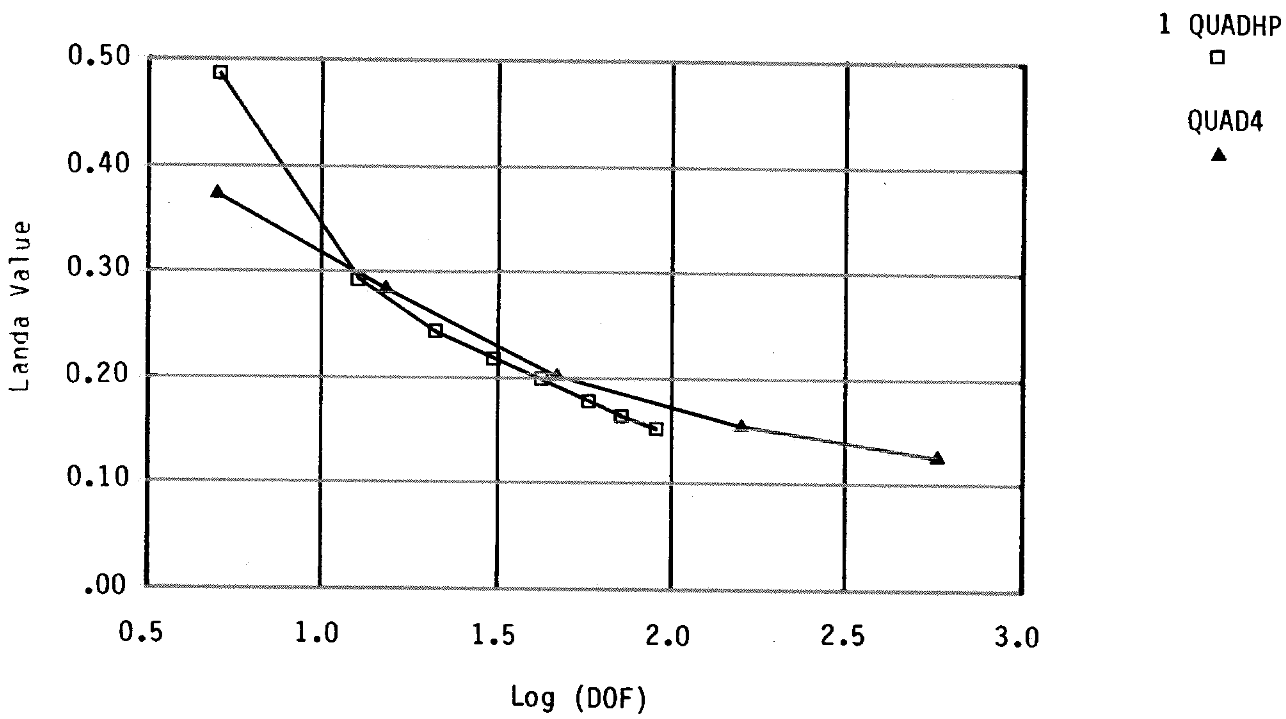


Figure 9. First Eigenvalue of Simply Supported Square.

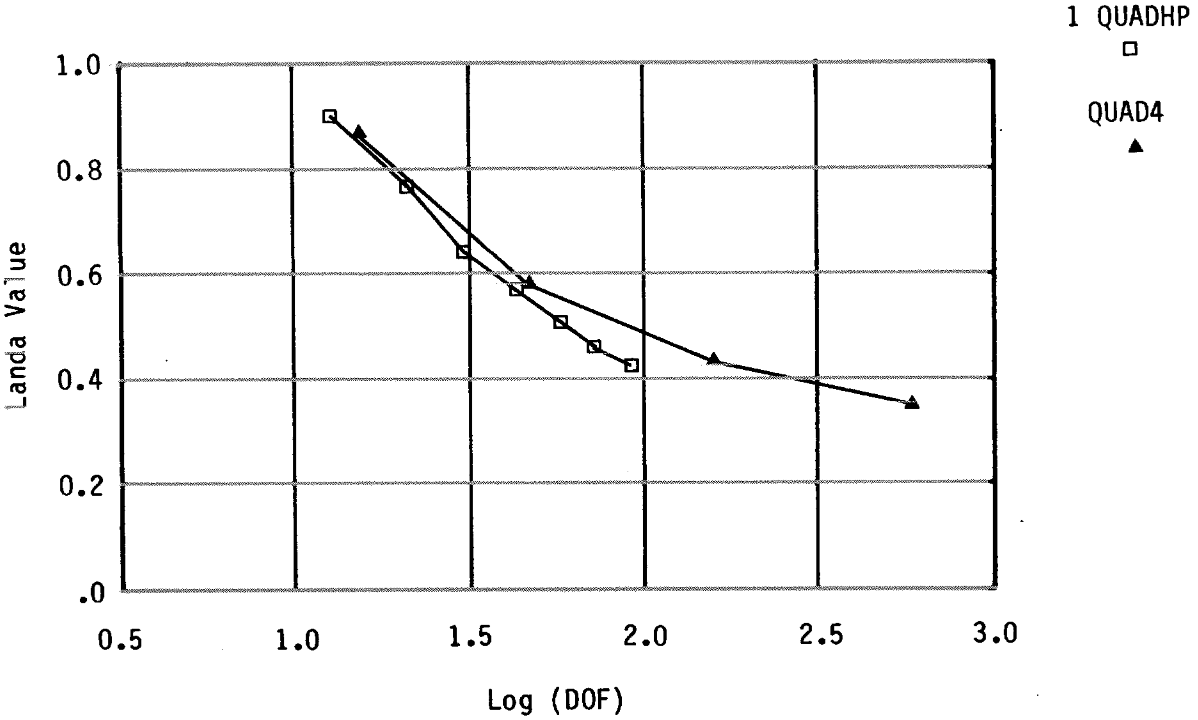


Figure 10. Second Eigenvalue of Simply Supported Square.

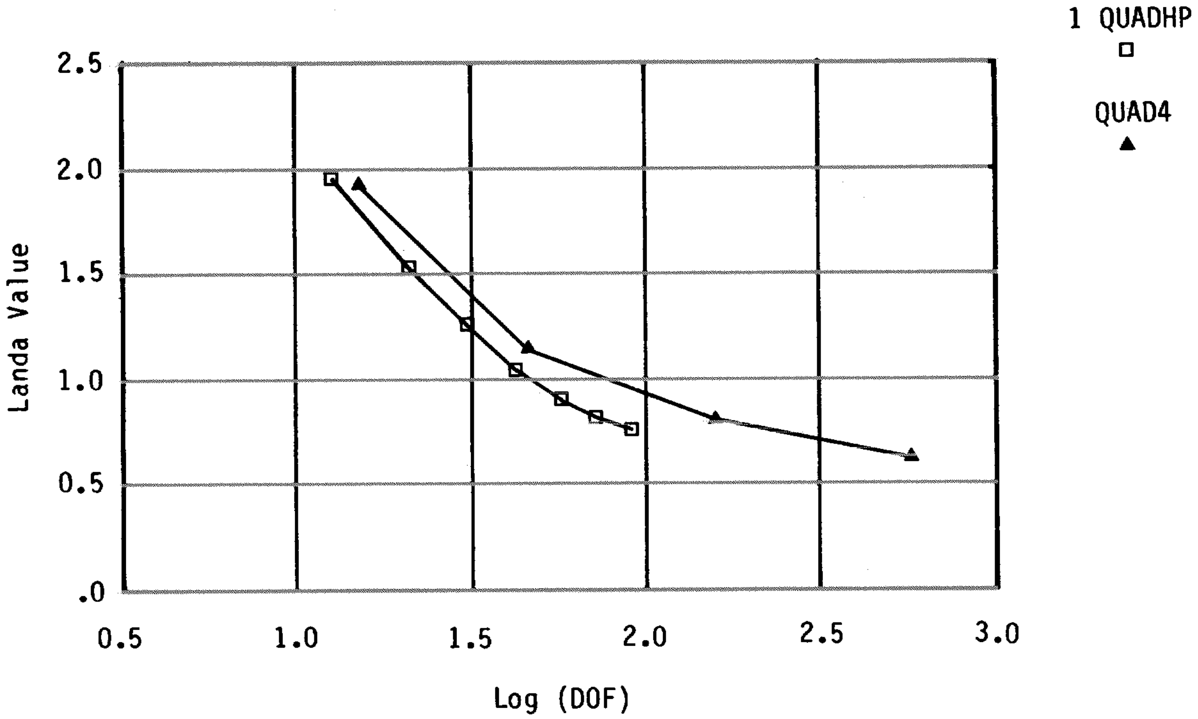


Figure 11. Third Eigenvalue of Simply Supported Square.

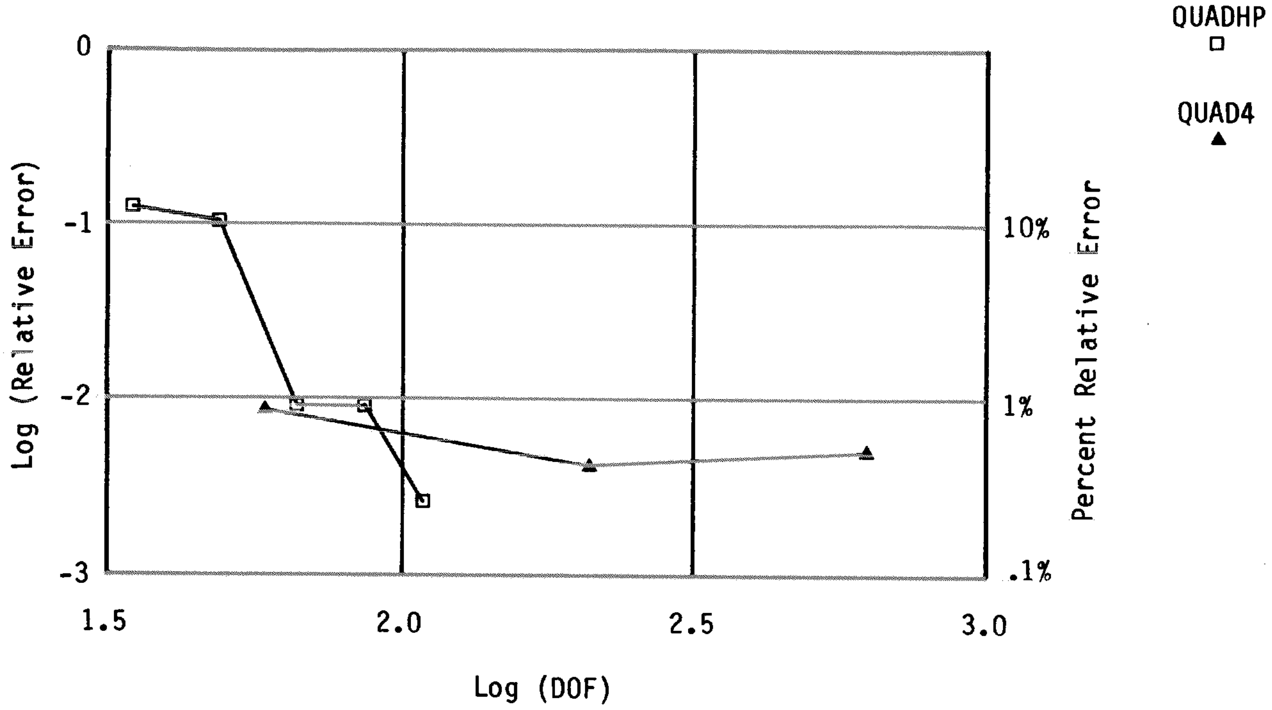


Figure 12. Relative Error of Displacement ω, at the Center of a 90° Simply Supported Plate.

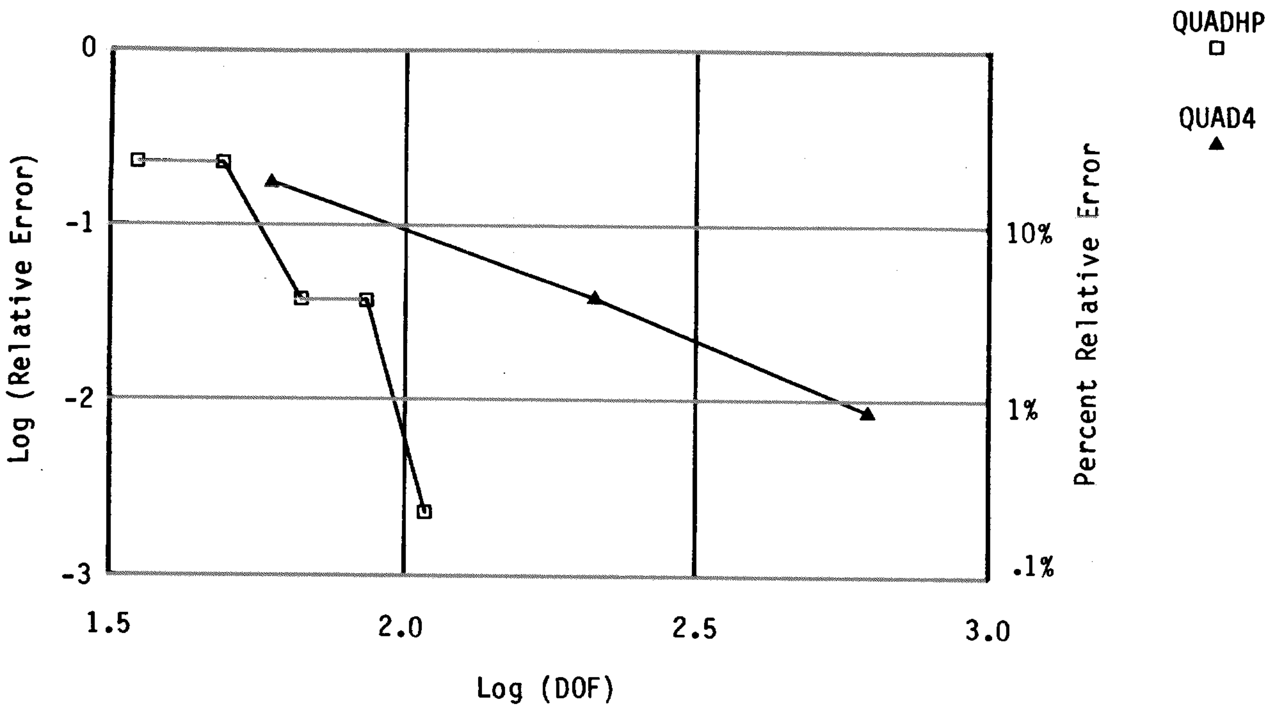


Figure 13. Relative Error of Maximum Moment Ma, at the Center of a 90° Simply Supported Plate.

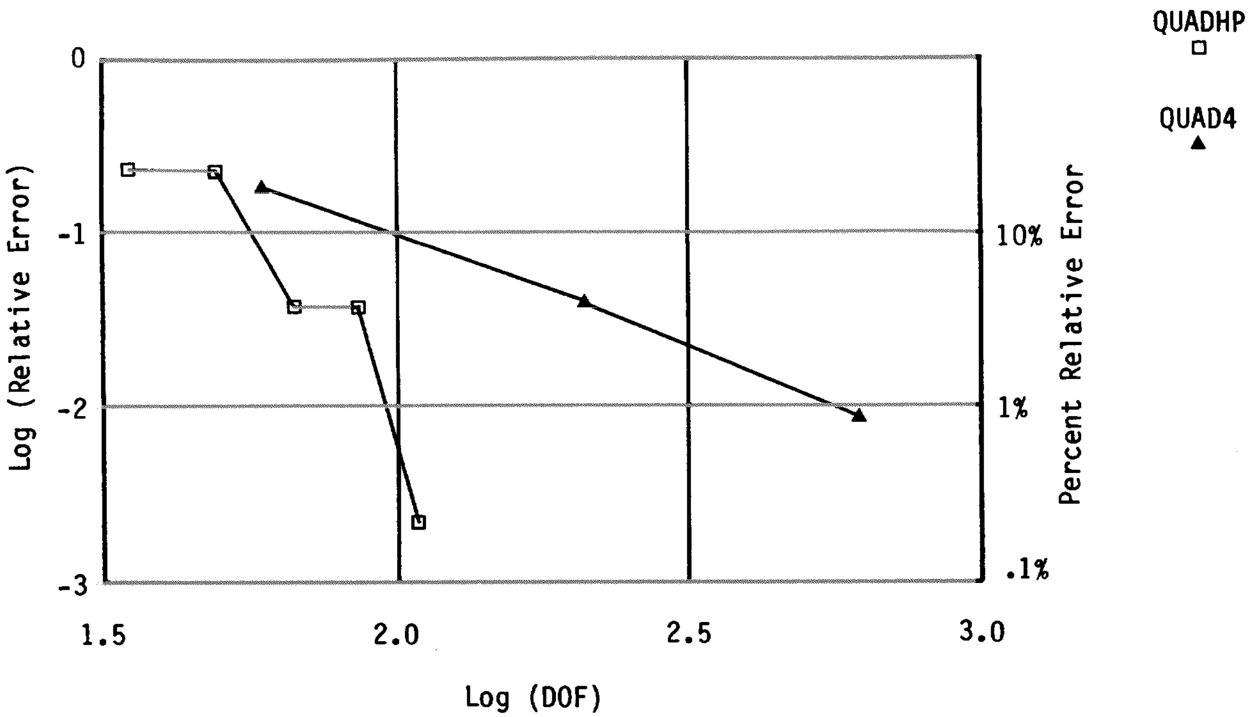


Figure 14. Relative Error of Minimum Moment Mb, at the Center of a 90° Simply Supported Plate.

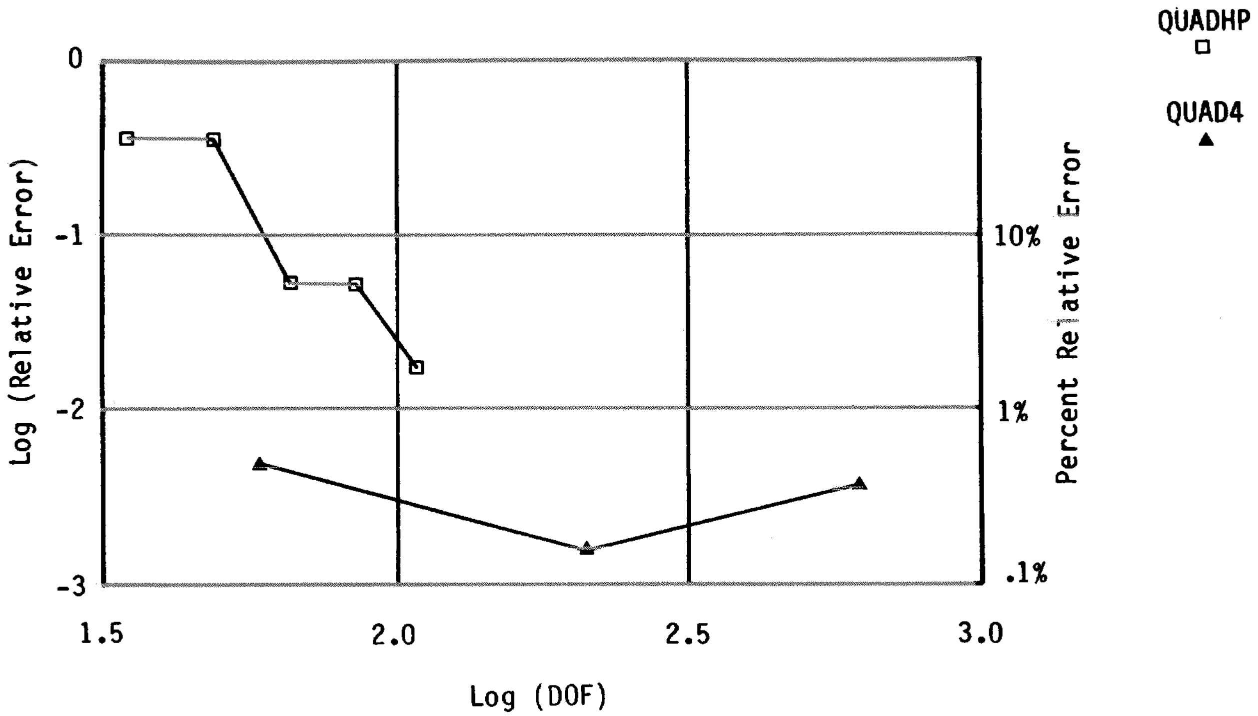


Figure 15. Relative Error of Displacement ω, at the Center of a 60° Simply Supported Plate.

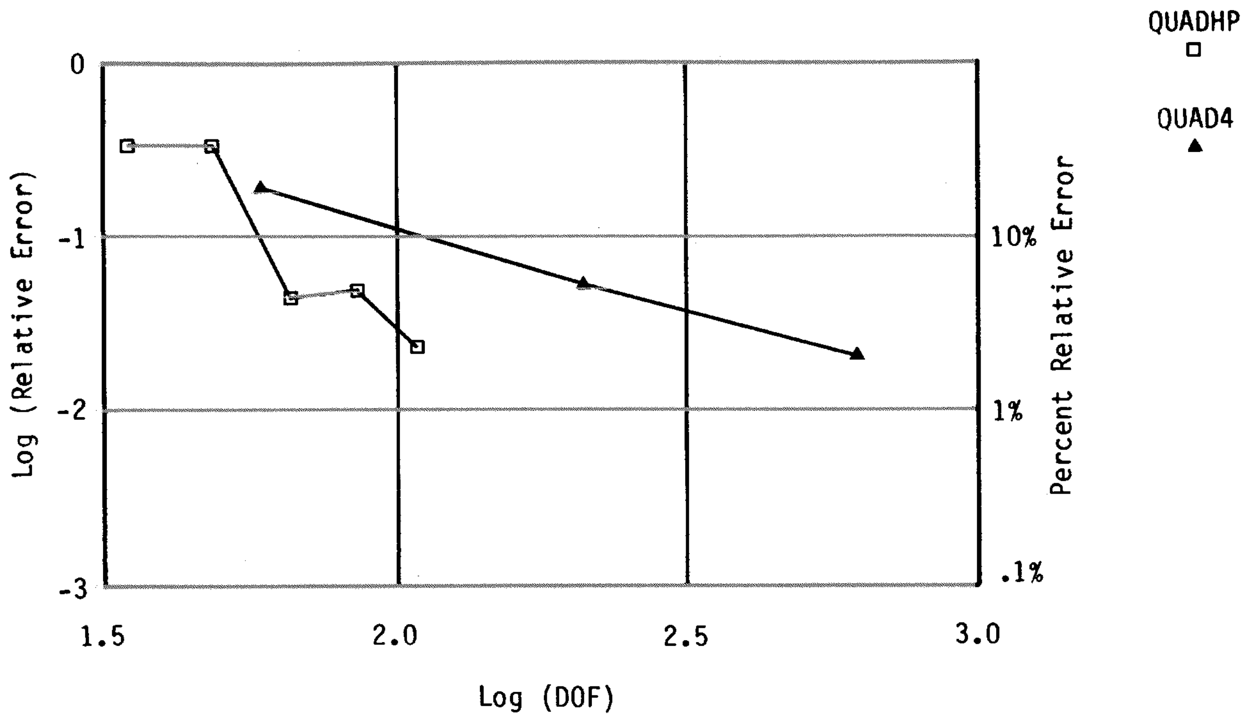


Figure 16. Relative Error of Maximum Moment Ma, at the Center of a 60° Simply Supported Plate.

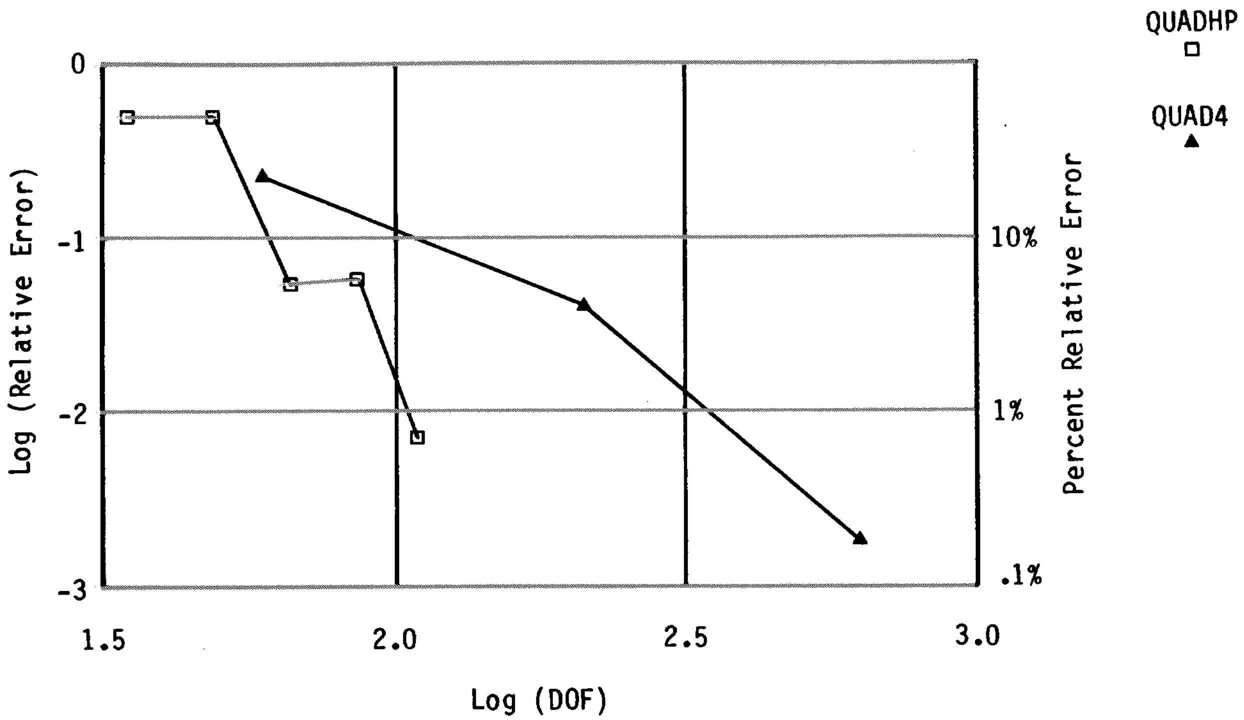


Figure 17. Relative Error of Minimum Moment Mb, at the Center of a 60° Simply Supported Plate.

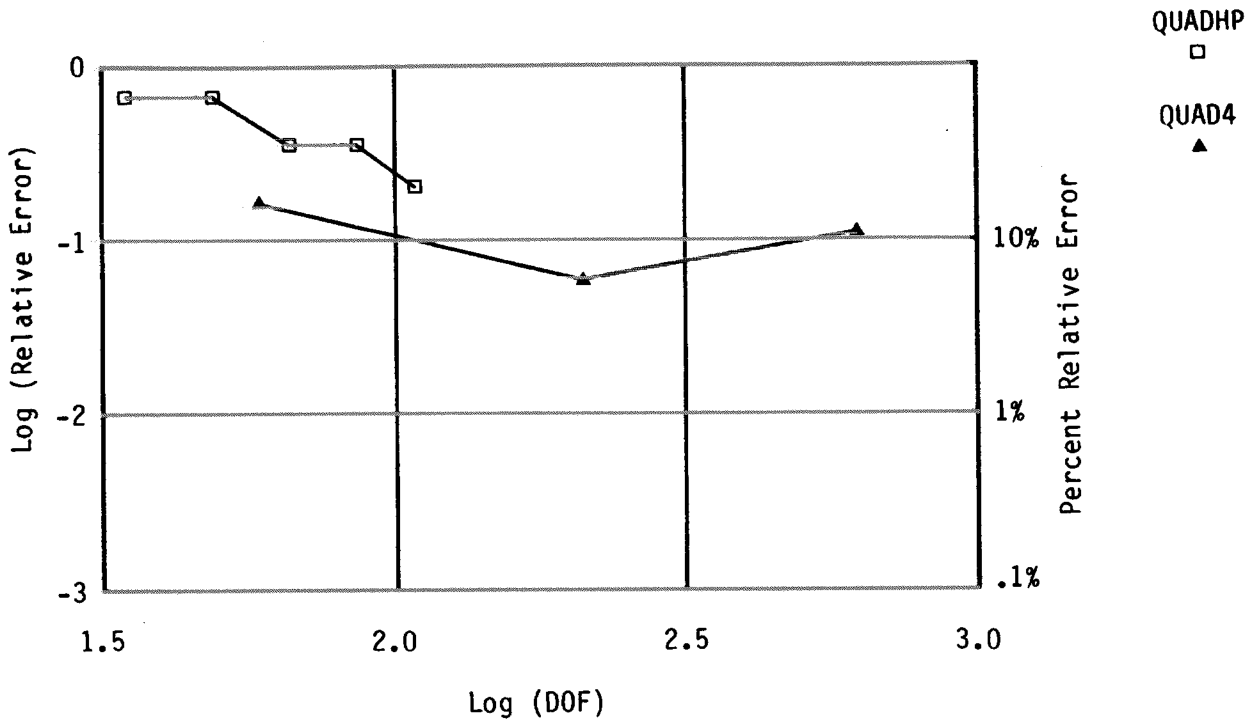


Figure 18. Relative Error of Displacement ω, at the Center of a 30° Simply Supported Plate.

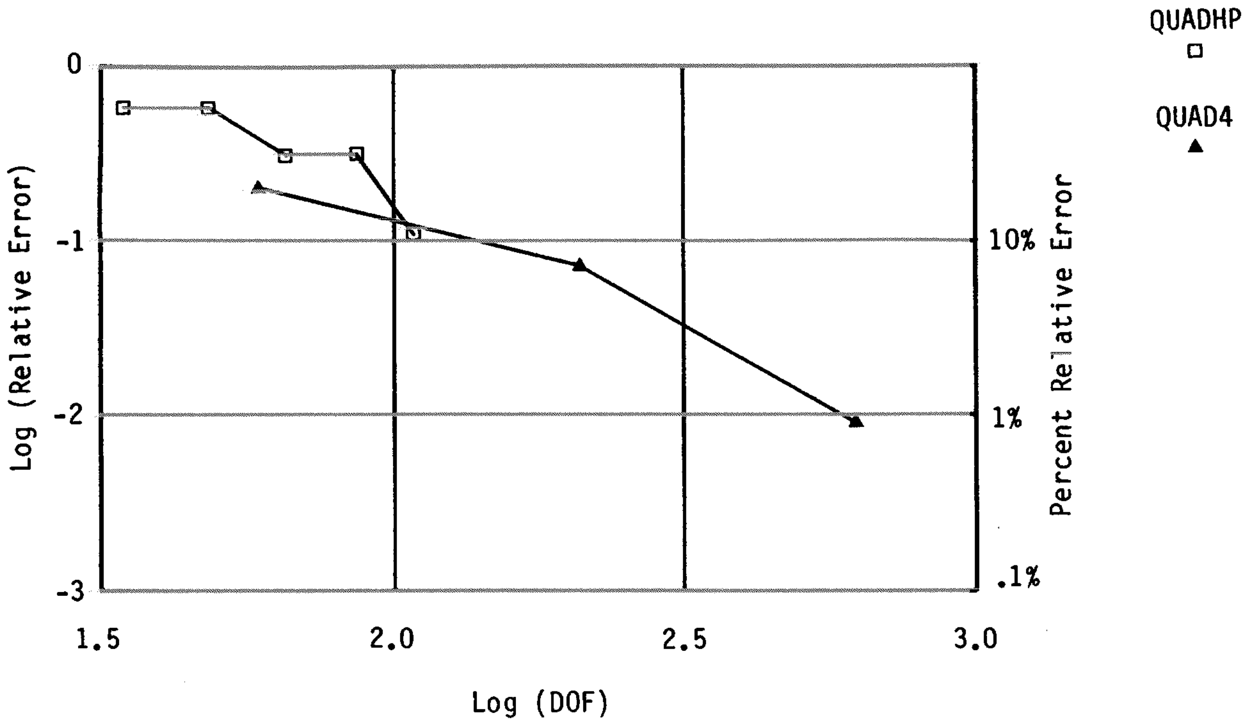


Figure 19. Relative Error of Maximum Ma, at the Center of a 30° Simply Supported Plate.

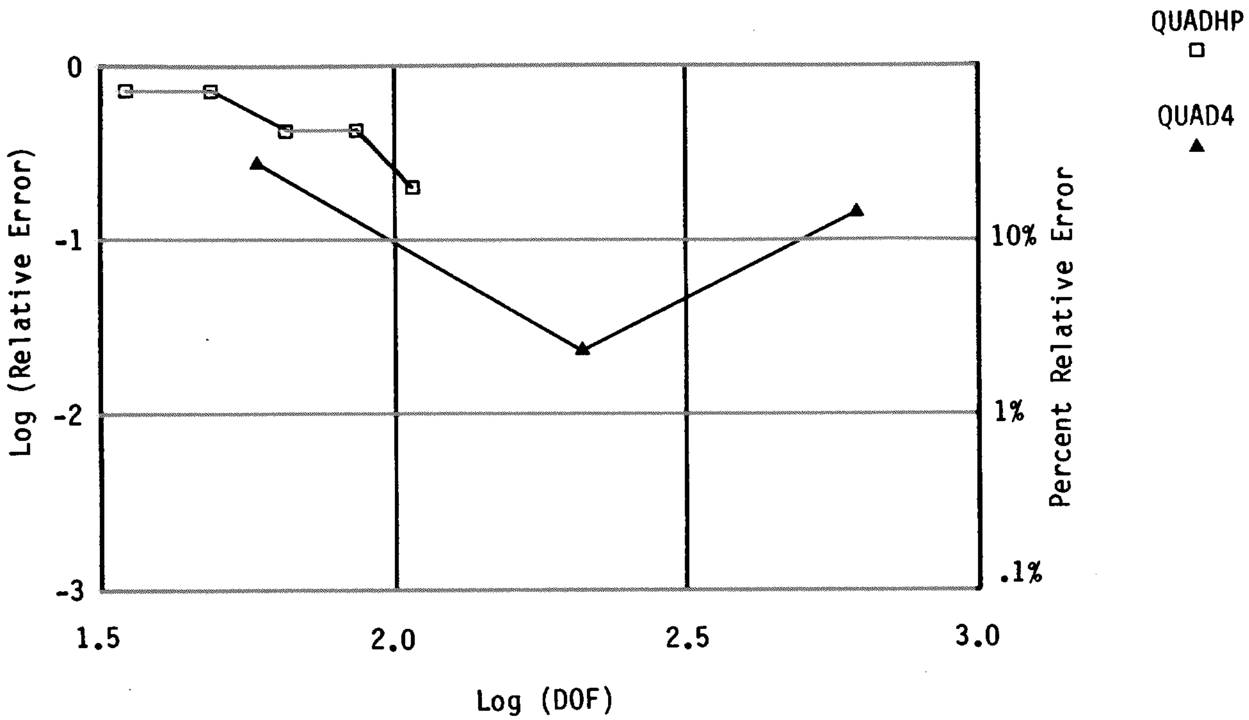


Figure 20. Relative Error of Minimum Moment Mb, at the Center of a 30° Simply Supported Plate.

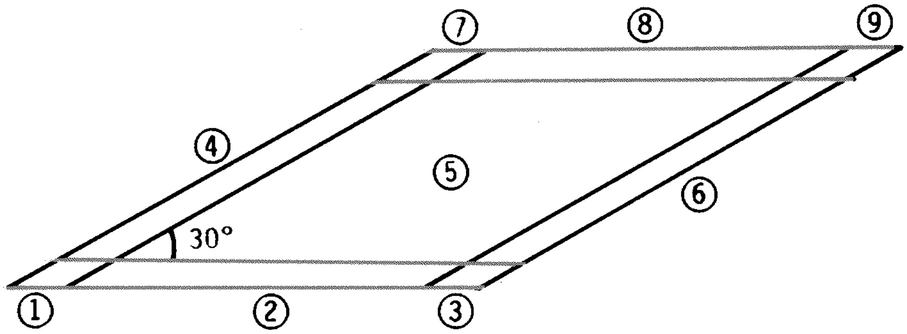


Figure 21. Refined QUADHP Mesh for 30° Rhombic Plate (9 element).

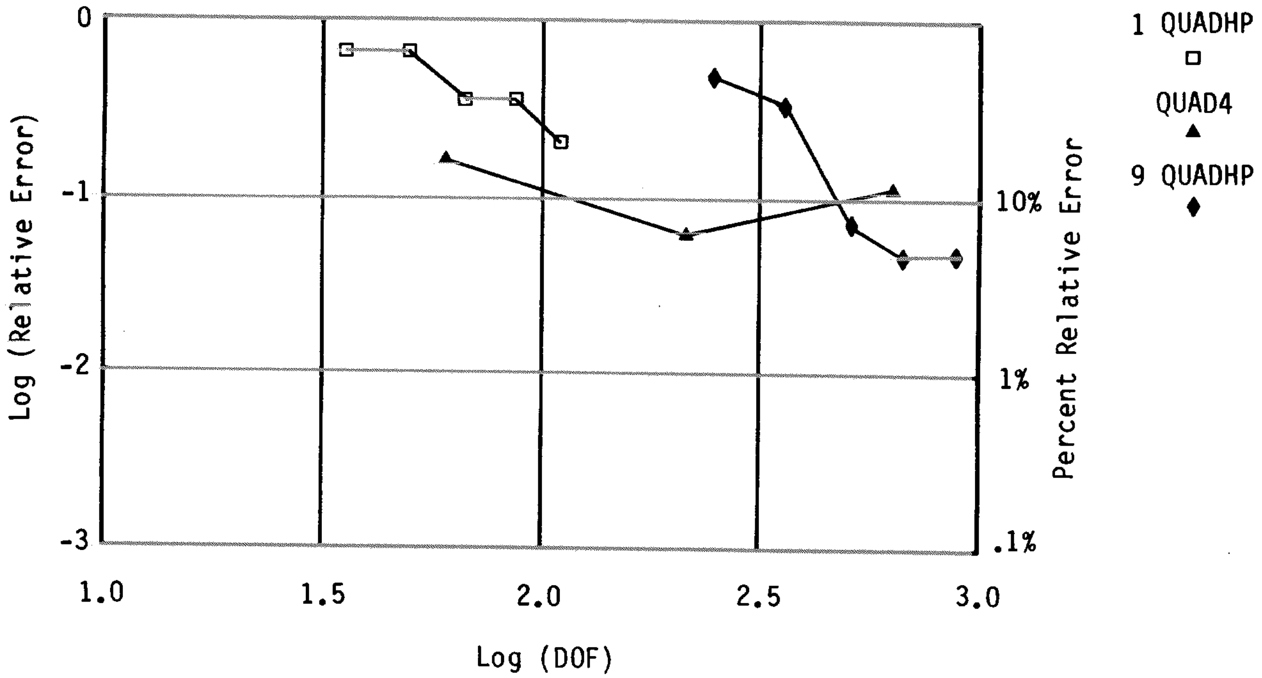


Figure 22. Relative Error of Displacement ω, at the Center of a 30° Simply Supported Plate.

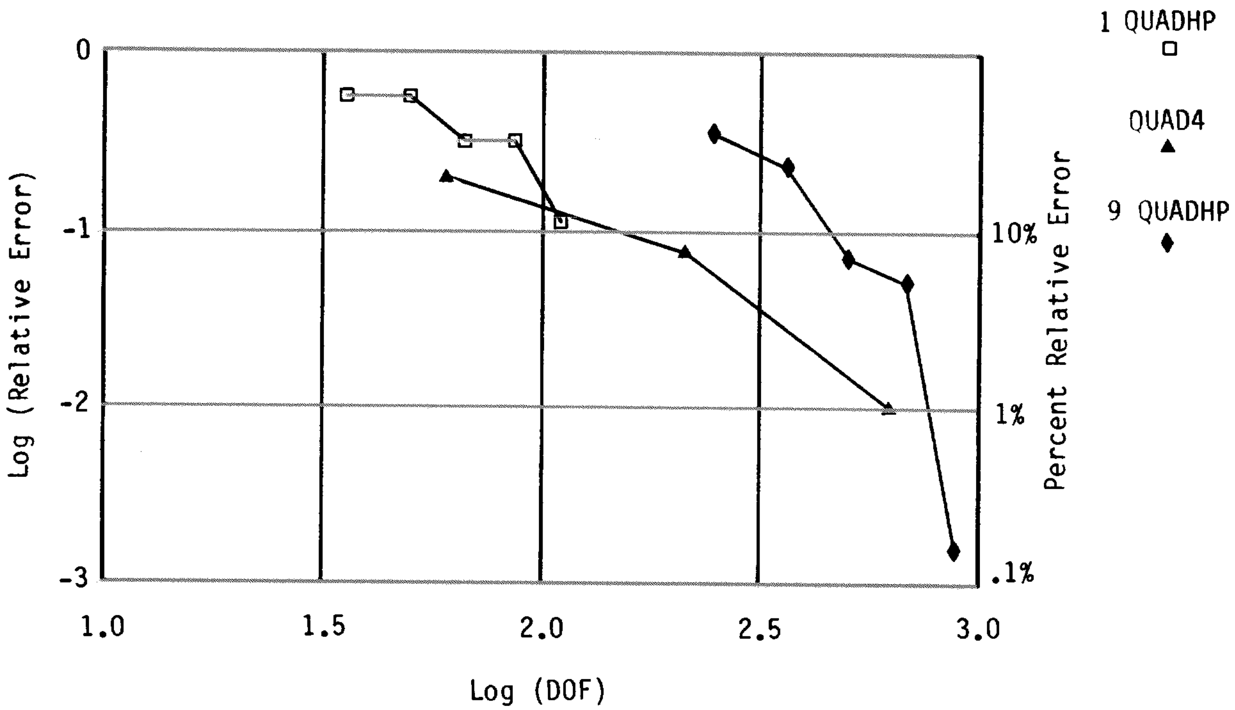


Figure 23. Relative Error of Maximum Moment Ma, at the Center of a 30° Simply Supported Plate.

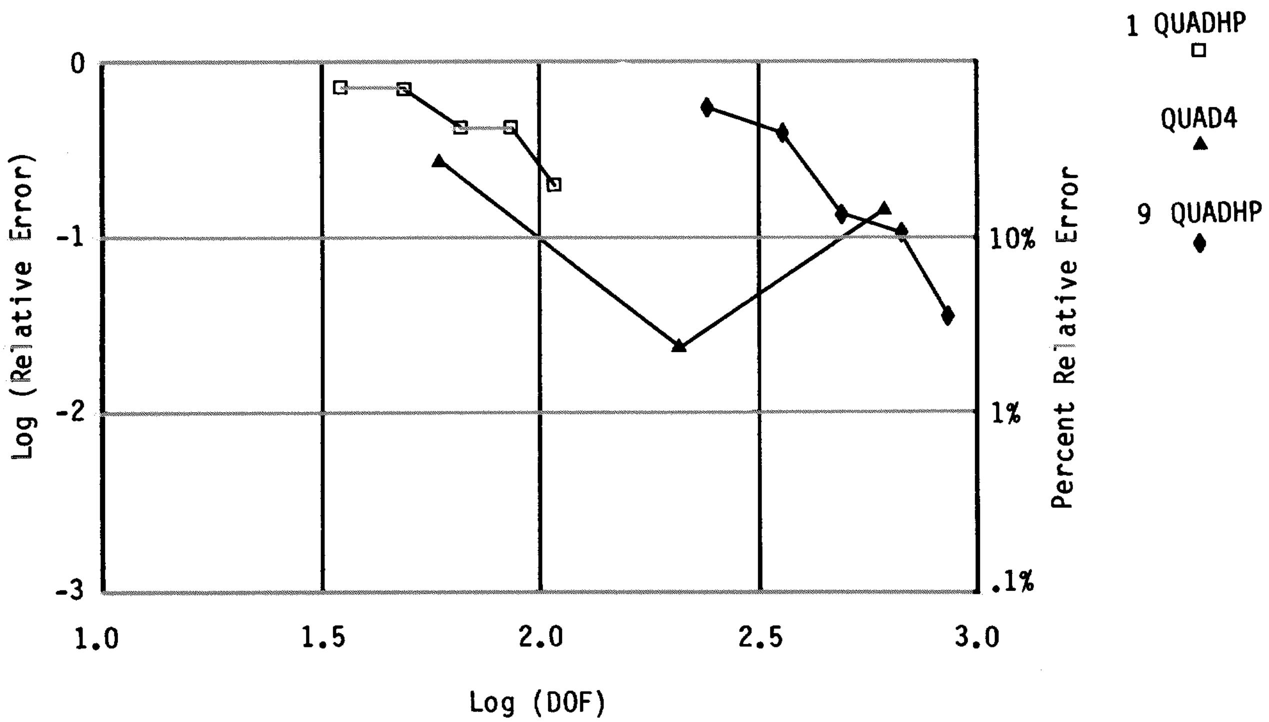


Figure 24. Relative Error of Minimum Moment Mb, at the Center of a 30° Simply Supported Plate.