B 8/6

A Comparative Study of Different Techniques in the Stress Analysis of a Nuclear Component

P.W. Dickenson, C.G. Floyd
Lloyd's Register of Shipping, 71 Fenchurch Street, London EC3M 4BS, U.K.

ABSTRACT

The inner surface stresses around the corner between the cylindrical wall and end plate of a flat ended pressure vessel have been determined using finite element, boundary element and photoelastic techniques. The results demonstrate severe deficiencies under certain conditions in the performance of the quadrilateral axisymmetric finite element which is commonly used in this type of analysis. The boundary element method is shown to provide an alternative analysis route giving more accurate results. The hybrid formulation finite element is also found to give reasonable results for the analysis of stresses in regions of rapidly varying stress.

1. INTRODUCTION

The use of direct numerical analysis techniques for engineering design is now commonplace. In particular, finite element methods are now routinely used in the stress analysis of nuclear pressure components, and the results produced are on occasions accepted without question and without confirmation from other theoretical or experimental approaches. This paper presents the results of a comparative study of different techniques used in the determination of stresses in a flat ended thick walled pressure vessel. The range of results obtained by the variety of methods used demonstrates the dangers inherent in the unquestioning acceptance of the results from a single analysis.

The objective of this comparative study was to investigate the accuracy and consistency of different theorectical and experimental techniques for a typical but demanding stress analysis problem. The problem chosen was the determination of the inner surface stresses around the small radius corner between the cylindrical portion and the flat end portion of a flat ended pressurised nuclear component.

The comparative study was split into two phases. The first phase included a photoelastic model of a simplified pressure vessel geometry together with two boundary element and three finite element models of the photoelastic model. This phase provided a direct assessment of the validity of the different theoretical analysis techniques. The second phase of the study consisted of several boundary and finite element analyses of a more exact representation of the pressure vessel geometry. The conclusions of the first phase of the study were then used to assess the accuracy of the results of the second phase analyses.

2. PHASE ONE STUDY

An axisymmetric photoelastic model of a simplified pressure vessel geometry was constructed using Araldite epoxy resin. Figure 1 shows the geometry and dimensions adopted. The model was loaded by an internal pressure of 2.6 psi. Results were obtained on a line passing though the cylinder wall from the point on the inner corner surface where the maximum principal stress occurred. This line is indicated in Figure 1. The variation of the three principal stresses along this line is shown in Figure 2. As expected, the major principal stress in the RZ plane is at a peak on the inner surface and falls to zero at the outer surface. The minor principal stress balances the internal pressure at the internal surface and then rises rapidly before steadily falling to a negative value at the outer surface. This also is as expected, because of the magnitude of the indicated bending stresses in the cylindrical wall caused by the pressure on the flat end of the vessel. The pattern of variation of the hoop stress with radial distance through the cylinder wall also reflects the dominance of the bending stress.

Three finite element models of the photoelastic model were constructed, using the NASTRAN, MARC AND ADINA programs. Each finite element model was constructed from axisymmetic elements; constant strain trapezoids and triangles for the NASTRAN program, and eight noded isoparametric quadrilateral elements for the other two programs. Linear elastic behaviour was assumed, and the models were constrained as shown in Figure 1. The photoelastic model material had a Young's modulus of 1435 psi and a Poisson's ratio of 0.5. These values were used in the theoretical analyses, although the value of Poisson's ratio was reduced to 0.4 to prevent numerical instabilities.

Two boundary element models of the photoelastic model were constructed, using the BEASY program and a program developed at Imperial College, London. For both programs three noded isoparametric axisymmetric line elements were used, and the models were loaded and constrained as shown in Figure 1. A Poisson's ratio value of 0.49 was assumed for the boundary element analyses because the numerical instability problems are less severe than for finite element analyses. The results for the two boundary element models were found to be virtually identical, and so only one set of the results is presented in this paper.

The final representation of the photoelastic model was constructed using the MARC non-linear hybrid formulation elements, assuming a Poisson's ratio of 0.5, and using approximate Mooney-Rivlin coefficients derived from the epoxy resin material properties. The element mesh, loading and constraints used were identical to those for the MARC linear elastic analysis.

The main results of all the above mentioned analyses are presented in Figures 3 to 7. As for the photoelastic model results presented in Figure 2, Figures 3 to 7 show the variation of the three principal stresses along a line passing through the cylinder wall from the point of maximum principal stress on the inner corner surface.

The results for the linear elastic finite element analyses are given in Figures 3, 4 and 5. All three sets of results show the same characteristics. Firstly, the major principal stresses on the inner surface are high when compared to the photoelastic result. This is not surprising, as finite element models are typically too stiff, leading to an overestimation of stresses. More worrying are the minor principal stress results, where the stress patterns close to the inner surface are clearly incorrect. One would intuitively suspect that these poor results are due to meshes which are too coarse, but tests using refined meshes did not show any significant improvement. Also noticeable in the finite element model results are the irregularities which occur in both the minor principal stress and hoop stress patterns. These irregularities are not due to any sudden change in mesh density, and their position does not change when a finer mesh density is used.

Unlike the linear elastic finite element analyses, the results for both the boundary element and the non-linear hybrid formulation finite element models, presented in Figures 6 and 7, show very good correlation with the photoelastic model results. These results would appear to demonstrate that the element formulations used in the linear elastic finite element analyses are incapable of accurately representing the rapidly varying stress field found in a component such as that modelled. This failing of the element formulation is not restricted to any specific program and is discussed in more depth in the following section.

3. PHASE TWO STUDY

The second phase of the study was an analysis of the actual component using finite element and boundary element techniques. The pump bowl was analysed using a common finite element idealisation for each of the finite element programs used. In this way it was considered that the differences within the programs due to the method of mathematical manipulation could be highlighted. Six finite element programs widely used in the nuclear industry were used in the project.

Phase One of the study clearly demonstrated that for this problem the results from the boundary element programs corresponded very closely to the results obtained from the photoelastic model. Therefore, a boundary element analysis of the actual pump bowl was performed to provide a standard for comparison with the results of the six finite element programs.

Discussion with the organisations marketing the finite element programs implied the suitability of using an eight node isoparametric axisymmetric element. The objective of the analysis was to determine the surface stresses in the corner between the cylindrical section and the flat end plate of the nuclear component. It was assumed that: the component could be modelled axisymmetrically; the effects of any outlet pipe forces were negligible on the region of interest; only part of the component and the inlet pipe need be modelled.

By choosing one type of finite element for use in all the programs under consideration, it was possible to use the same element mesh in each. Figure 8 shows the mesh chosen. The model was constrained at the half length of the component and inlet pipe. The type of constraint used was dependent on the program, but essentially the component end was allowed radial freedom only. The model was loaded with internal pressure and an axial inlet pipe reaction force. Steel material properties were assumed.

The major and minor principal stress distributions around the corner surface were selected as the two main criteria for comparing the results of the different programs. Surface principal stresses were selected, primarily because they are the most important in engineering analysis, but also because they are determined in finite element analyses by extrapolation from the element Gauss points and hence are liable to greater inaccuracy. The value of minor principal stress on the corner surface should equal the applied internal pressure, and this value therefore provides an additional check on accuracy.

Figure 9 show the scatter in the major principal stress results produced by the various finite element programs. It also shows the significant deviation of the minor principal stress results from the expected constant value commensurate with the pressure load. The reasons for this deviation have not been established, but it is suspected that it is due to a fundamental limitation in the mathematical formulation of the standard axisymmetric element. It is interesting to note that the pattern of deviation is consistent for all the finite element programs used.

Subsequent to the initial comparison work, Lloyd's Register mounted the MARC finite element program on their in-house computer, thus enabling a more detailed study of this program's performance. Figure 10 shows the MARC results around the corner surface, using the mesh shown in Figure 8. In order to assess the effect of refining the mesh in the areas of interest, a second mesh was produced as shown in Figure 11. The results from this new mesh are plotted in Figure 12.

The MARC element library contains elements developed for modelling incompressible of nearly incompressible materials, and such hybrid formulation elements can also be used for the analysis of compressible materials such as steel. Figure 13 shows the results obtained using these hybrid elements. It can be seen that this element formulation has improved the results for the analysis, but these still do not match the accuracy of the boundary element results.

4. CONCLUDING REMARKS

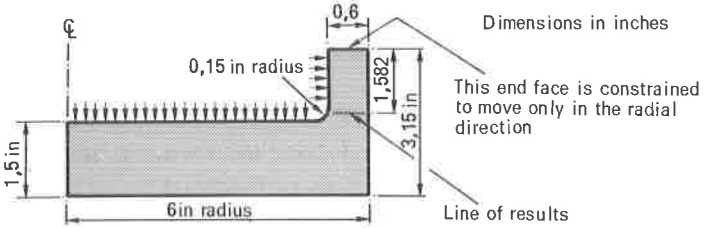
The main purpose of this paper is not to provide a detailed comparison of different analysis programs, but rather to illustrate the dangers of unquestioning acceptance of the validity of any analysis program in complex stress situations. A finite element is normally tested in a range of simple models before being released, and it is suggested that perhaps such models are not sufficient to provide full verification of the element performance. In particular, the results presented in this paper show that the finite elements used do not provide adequate results in regions of rapidly varying stress. Neither does detailed mesh refinement ensure more accurate results, as numerical instabilities can occur. For the problem discussed in this paper, a change in element formulation led to significantly improved results, indicating that the poor results for the isoparametric elements are due to a general failing of the isoparametric axisymmetric element formulation.

One of the main sources of error in this type of finite element analysis is the extrapolation of stress results from the element Gauss points to the corner nodes. The range of results obtained for the same element formulation used in the same mesh indicates significant variations between the programs in the numerical methods used, including the Gauss point extrapolation routines. The possibility of inaccuracies in the numerical methods used in finite element programs, combined with doubts about the validity of individual element formulations, highlight the dangers inherent in the indiscriminate use of finite element methods. Furthermore, these dangers are inherent in the use of any theoretical analysis method. Although the boundary element programs gave good results for the particular problem analysed, partly because of the direct calculation of the surface stresses in such programs, it cannot be assumed that the programs will perform well under all conditions. Accuracy of results can only be assured by cross checking against results obtained by a different analytical route.

ACKNOWLEDGEMENTS

The results presented in this paper are taken from a research project carried out by Lloyd's Register of Shipping and the authors wish to acknowledge the contributions made by their colleagues, in particular Dr Khayyat for his photoelastic work. The views expressed in the paper are those of the authors and not necessarily those of the Committee of Lloyd's Register of Shipping. The authors wish to thank their head of department Mr D. Rennie for his encouragement and advice, and the Chief Engineer Surveyor, Mr C. Archer, for permission to publish.

Fig 1



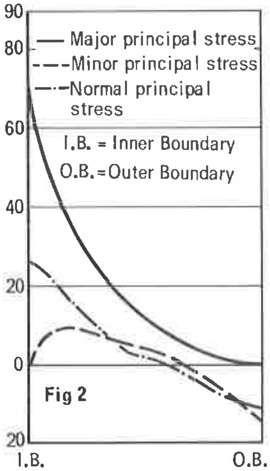


Fig 2

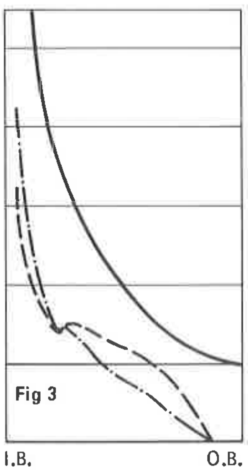


Fig 3

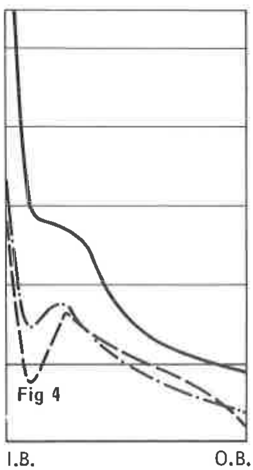


Fig 4

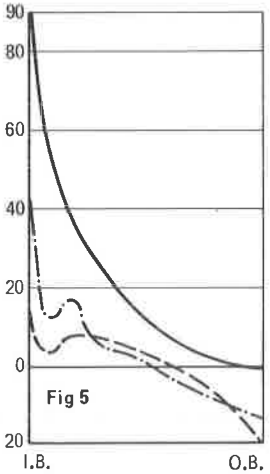


Fig 5

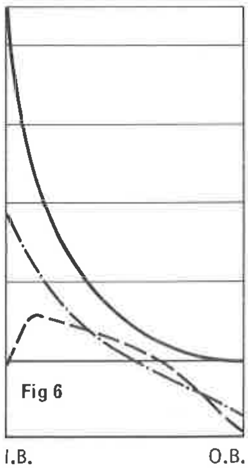


Fig 6

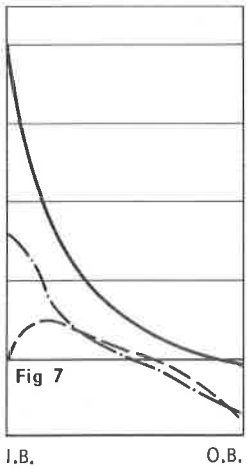


Fig 7

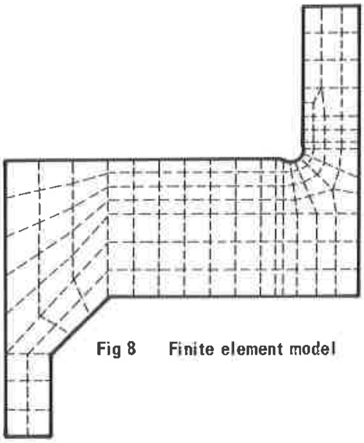


Fig 8 **Finite element model**

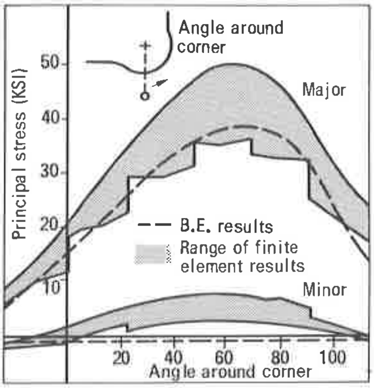


Fig 9

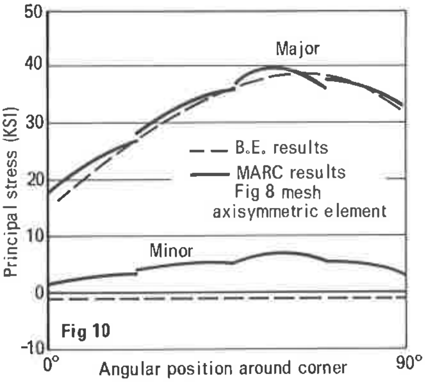


Fig 10

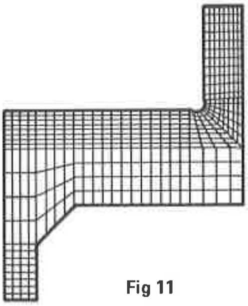


Fig 11

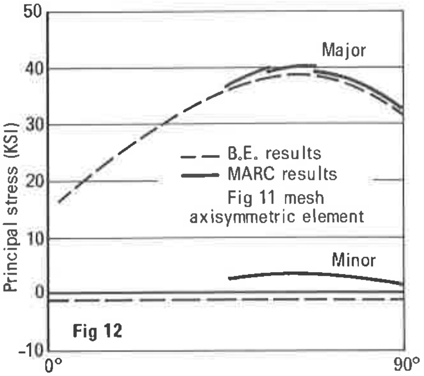


Fig 12

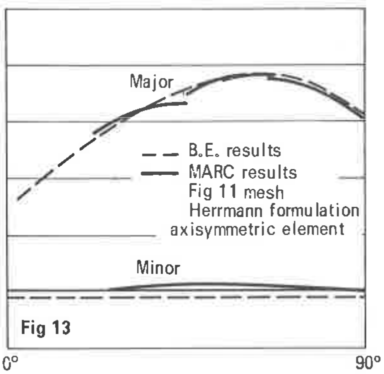


Fig 13