THE APPLICATION OF THE "BERSAFE" FINITE ELEMENT SYSTEM TO NUCLEAR DESIGN PROBLEMS

T.K. HELLEN,

Central Electricity Generating Board,
Berkeley Nuclear Laboratories, Berkeley, Gloucestershire, United Kingdom

ABSTRACT

BERSAFE is a general computer system designed to perform the stress analysis of arbitrary two-dimensional and three-dimensional structures using the method of finite elements. The system is currently available for elastic analyses for a large number of different element types, which include plane stress, mathematical and engineering plane strain, axisymmetric and solid three-dimensional structures. Different kinds of loading may be applied, including point loads at nodes, line and pressure loads over edges and faces, body forces, rotational forces and temperature distributions. Degrees of freedom may be restrained from moving or given prescribed values, and a decoupling facility is available allowing cracks to open in tensile regions. Material properties may vary from element to element and be temperature dependent.

Auxiliary routines are available for the automatic generation of meshes for all two-dimensional and certain three-dimensional structures, and computer plots may be obtained for stress contours in two-dimensional cases and deformations in all cases.

The paper describes the facilities and element types available, together with reasons for adopting a system approach. Examples of various applications which have been analysed by the system are described. These include a three-dimensional analysis of a podded boiler prestressed concrete pressure vessel containing holes and subjected to internal pressure. The problem of the three-dimensional cylinder-cylinder intersection is also described, including methods of mesh designing and comparisons with experimentally-obtained data. Finally the use of solid three-dimensional elements as thin shells is described including their performance for thin, pressurised cylinders with end restraint, and the wind-loaded cooling tower problem. These indicate the usefulness of the solid elements as thin shells, particularly in plant problems where transitions from thin shells to thicker areas are required and full element compatibility is to be maintained.

1. INTRODUCTION

BERSAFE is a general computer system designed to perform the stress analysis of arbitrary two and three-dimensional structures using the method of finite elements. The structure may be subjected to point loads, line loads, pressures, body forces, rotational forces and temperature distributions. Any parts of the structure may be fixed or allocated prescribed movements, and a facility exists for openings to arise in tensile fields to.
simulate crack openings. Material properties may vary throughout the structure and be temperature dependent.

The system has been developed for the solution of plant problems arising in the C.E.G.B. and has been implemented on their IBM S/360 model 85 computer. Although a certain amount of effort has been expended in achieving efficient running times on this particular configuration, the system is still capable of being transferred to other computers with only minor changes. The programming language used is Fortran IV.

In the finite element method the structure is represented by an assemblage of small areas or volumes, known as elements, each having some prescribed variation in all field variables, such as stress, displacement and temperature. Thus, if constant stress elements were being used, a large number of them would be required to simulate areas of high stress gradients. The corners of elements are termed 'nodes' and act as reference points for specifying geometry, loading and other input data, and for presenting output values. Nodes may also appear at intermediate positions along element sides, which may then be curved. Such elements thus contain more nodes, but less of them are required to achieve a required accuracy than the more simple elements. The essence of the finite element method is to determine all the element stiffness matrices, superimpose them to obtain a structural stiffness matrix, then obtain the resulting deflections using a formula obtained by applying the minimum strain energy criterion. A large number of types of element are available, each having prescribed properties. Different types of element may be freely mixed within one structure, although the user should ensure that the mixing is physically sensible.

Phase I of the system development calculates elastic displacements and stresses at the nodes, with extra optional output for element values. Further output options include the reactions corresponding to prescribed displacements and principal stresses. Subsequent phases of the system will deal with dynamics and inelastic analyses, including large deflections, plasticity and creep.

An inherent disadvantage of the finite element method is that a large number of nodes are required to represent real structures such that reasonably accurate results are obtained. The input data specifying the geometry and the topology, which is a list of the nodes lying on each element, is inevitably large in quantity and has to be accurate. Consequently, the system checks all input data to remove as many errors as possible. Further, two-dimensional and some three-dimensional structures may be generated automatically using mesh generation routines. For these, the user specifies a few boundary points which surround areas containing large numbers of elements which are reasonably similar in shape and size, and the routine actually generates topology and geometry data for those elements, and passes the data straight into the stress analysis step. Several such areas may be joined to form all or part of the total structure. This facility can reduce the quantity of input data drastically, and so saves preparation time and avoids input errors. An alternative method of data preparation is available in the form of a DMAC digitising table, a device which produces punched cards from a mesh drawing by using a scanner. The facility is particularly useful for those meshes having complex boundaries and rapidly varying elements, and again is fast and accurate.

When the input data, however obtained, has been fed into the system, a final check on
the accuracy of the structural representation may be obtained from a plot of the mesh. Using a Calcomp plotting table, any errors in topology and geometry are readily detected. This device also produces plots of the deformed structure, and stress contours for certain two-dimensional cases.

Prior to performing a stress analysis of a given structure, it may be desired to perform a thermal transient analysis. The program FLHE (Fullard [1]) has been developed for this purpose and is completely compatible with BERSAFE. The same geometry and topology input formats exist, and this data does not have to be resubmitted to BERSAFE after a FLHE analysis.

2. THE SYSTEM APPROACH

The advantages of developing computer systems for finite element techniques, instead of a number of programs each performing limited analyses, have been realised for a considerable period of time. The system approach involves having a minimum number of program routines, or modules, to enable any given analyses within a general framework to be carried out. For instance, the only difference between a two-dimensional and three-dimensional analysis lies in the particular element matrix modules used. Early system developments were very much restricted because of limited hardware and software facilities then available and the limited technical knowledge. However, the latest generation of computers in recent years has enabled the evolution of effective systems.

The ideal system is a computer package which enables both the user and the researcher to solve any conceivable problem or incorporate any new developments efficiently and with minimum effort. However, the attainment of such a system is practically impossible due to the inevitably high cost, current computer hardware and software limitations, the difficulty required in using the system, and possibly a loss in efficiency in such areas as solution technique and data storage on files. All existing systems are a compromise depending on the individual requirements. It is extremely important to plan carefully prior to development, including the assessment of how much effort and money can be expended, and to define exactly the scope of the system. In early development stages it is often difficult to plan to the required specification, so a certain flexibility in the development stage should be allowed. A common problem at this stage is meeting estimated completion dates, which underlines the mistake of incorporating too many facilities in early phases. The final product must be efficient, well tested out, and easily usable and understandable, otherwise the entire project could be a financial disaster.

Probably the most important aspect of a production system is the efficiency of the interface between it and the user. A poorly designed interface is the result of rapid programming, and is only adequate if the sole user is the programmer, which defeats the object of having a system available to a large range of users. The interface should be designed to be easily understood by any practising engineer who has received some basic training in the concept of finite element techniques. The interface can be broadly categorised into two groups:

(a) a steering language, or agenda, which is a sequence of executable program statements defining a course of computation through the system, with associated input data as and where necessary, and

(b) an inference technique, which is possible for systems having courses of computation
which lie within certain bounds. A certain course is usually assumed, and any deviations are indicated when necessary. Only input data is required.

The steering language is some high level language containing statements which invoke the various elementary operations of the system, such as solution, or evaluation of element stiffness matrices, and the statements may vary greatly in scope of operation. Matrix handling schemes may be incorporated, and even entire programming languages such as Fortran or PL/1 may be available. Individual types of analysis, for instance a stress analysis of a two-dimensional structure, is possible using one particular set of statements of the steering language, and would require some pre-defined set of actual input data. Such a system is easily extendible and very convenient for use in research, but can be difficult to use in everyday engineering applications because of the necessity of writing a steering program each time, unless special packs are made available for each type of application. A further disadvantage is that input data tends to be rather complex, being associated with a large number of different operations in the steering language.

The inference technique is a more convenient method to use in the application area. The system is designed to perform only one broad function, within which all the expected individual features are made available. Thus, any elastic and inelastic stress analysis system using finite element techniques can be designed using an inference technique but extensions to perform, for instance, arbitrary matrix operations require extensions to the actual system. The most likely course of operation is assumed and any changes or basic data for this are defined on input data. Careful planning is necessary in designing the input data so that full generality is available and the input is easy to use. Badly planned input can contain many indicators of which data cards should be presented and which should be omitted, which can become very confusing and give rise to data errors. However, when the input is carefully designed, the inference technique becomes a very attractive proposition, and the user need only learn one minimal set of input rules even if he is concerned with several different applications. New features in the system may be added with minimal input changes, and improvements due to theoretical developments may be incorporated without the user being directly concerned.

The BERSAFE system uses an inference technique. Phase I of the system has been written for the solution of elastic problems for arbitrary geometries, with a large number of features and element types for which the inference technique is quite adequate. The auxiliary operations of mesh generation, result plotting and thermal transient analyses are also available on an inference basis and are efficiently used in conjunction with BERSAFE with a minimum amount of input data. The inelastic extensions planned for subsequent Phases of the system will also incorporate an inference technique.

The facilities and element types available in Phase I have been carefully planned to cover the majority of envisaged stress analysis problems in C.E.C.B. plant. It has not been necessary to incorporate thin plate and shell elements at this stage since most thin-walled plant applications are axisymmetric and can be analysed using a parallel finite difference system PATAS (Goodman [2]).

3. SOLUTION TECHNIQUE

In BERSAFE, the finite element displacement method is used giving a set of equations
to be solved in the form:

\[ [K] \{\delta\} = \{\lambda\} \]

where \([K]\) is the structural stiffness matrix, \(\{\delta\}\) is a vector of unknowns and \(\{\lambda\}\) is the corresponding vector of point loads. The equations are solved by a direct solution of simultaneous equations, using a computer algorithm called a front solution (Irons [3]; Hellen [4]). This algorithm takes advantage of the banded nature of the \([K]\) matrix and also its positive definite, symmetric form. In the forward elimination, each equation, which corresponds to one degree of freedom in the structure, is eliminated at the earliest possible moment and in this way a minimum amount of the structural stiffness matrix is required on computer core at any one time. Only fast core is used in the actual elimination process since the use of auxiliary core involves very large computation times. Since the amount of fast core is limited, so is the size of the minimum structural stiffness area, which in turn limits the size of the band of this matrix. Because of the symmetric nature of \(K\), only half of the band is used, and the maximum size of this band is called the semi-bandwidth. This value represents the maximum number of degrees of freedom across the structure at any stage in the elimination process, and must not exceed 228 in Phase I of BERSAFE. The larger the semi-bandwidth, the larger is the computation time for a given number of equations. The actual number of equations solved is in theory unlimited as long as the maximum value for semi-bandwidth is not exceeded, although the total number of equations in BERSAFE is limited by the number of nodes which may be used. The semi-bandwidth is dependent on the ordering of topology cards. On these each element is defined on one card as consisting of a set of nodes, given in list form in some prescribed order, fixed for each type of element. This order should be such that the appearance of each element is progressive through the structure in rows, such that the largest row is as short as possible.

4. FACILITIES AVAILABLE IN BERSAFE

The general nature of BERSAFE requires that a large number of facilities and element types be available. Each user problem only requires a relatively small number of these options. The input data cards have therefore been designed so that the user specifies exactly what he requires and no more. Those facilities which are always required need not be defined in any way unless specific data is needed. A summary of the facilities available is given below.

As the input data cards are read by the system, they are checked for correct sequencing order and the individual contents are verified as being reasonable quantities. As much as the input is checked as possible, even if serious errors have already been detected. Each part of the system is only executed when no data errors have been detected in the previous part.

Geometry and loading data may be specified in cartesian, polar or cylindrical coordinates, irrespective of element type used. However, the direction of prescribed deflections and decouplings depends on the element type being used. Output stresses may be obtained in cartesian, polar or cylindrical coordinates, again irrespective of element type, and principal stresses, with the Von Mises equivalent stress and principal angle or direction cosines for three-dimensional cases, are available. The maximum number of nodes and elements allowed is 2000 and 1500 respectively, although the element number limit decreases as more
complex elements are used.

The decoupling facility enables any number of the degrees of freedom at any node to become independent between different elements meeting at that node. Typical cases are in crack openings and shear slip problems.

A variety of different loadings may be applied to any part of the structure. These include point loads, line loads and facial pressures, although the last two are not available for simple elements. Centrifugal loading may be applied to the entire structure about axes dependent on the element type, and body loads may be applied in any of the cartesian directions. Nodal temperatures may be specified to induce thermal strains relative to some datum temperature, either from nodal geometry input cards or from any required time from a preceding FLHE temperature transient analysis. Constraints are defined by specifying up to 650 displacements as zero or some prescribed quantity. The corresponding reactions may be calculated.

Three independent stressing cases may be applied for the same run. Each case can have any type of loading independently of the loading of the other cases, except temperature loads, which are common to all cases.

The plane strain facility available for two-dimensional structures is divided into two separate conditions, mathematical plane strain and engineering plane strain. In the former, no movement in the out-of-plane direction is allowed, and so out-of-plane stress components exist. The condition is easily obtained by modifying the material properties from the plane stress values. The latter allows an out-of-plane strain to exist in order that the out-of-plane facial forces are zero. The solution for this case is more involved and only one stressing case is allowed.

All material properties are specified on separate input cards of the same type. Several cards with the same material may be given if temperature-variable properties are required. Then, the element temperature is deduced from the nodal temperatures and an interpolation or extrapolation for the correct material properties is carried out. Cards of the same material with several temperatures specified must be grouped together in increasing temperature order. If only one temperature is given, the material properties are assumed constant for all required temperatures.

To reduce the amount of input data the user has to supply, default values may be given for the main element type, the main material and the main thickness for plane elements. These values are specified on the job information card, which also defines computational options to the systems. Deviations from the default values are specified only when necessary.

Plots of all meshes may be obtained using a Calcomp plotting table in order to check the correctness of the input topology and geometry data. The plots are drawn before the stress analysis part of BERSAFE so that the latter need only be conducted when the input data has been verified to be correct. Several plots may be requested in one run, each representing a view from a different point in space. If two plots are requested with view points suitably close together, a stereoscopic pair may be obtained for use with a stereoscopic viewer.
After completion of the stress analysis, plots of the deformed structure may be obtained. For two-dimensional structures, contour plots of any stress components may be obtained.

The available mesh generation routines may be used for any two-dimensional structure, by specifying only broad outlines on the input data. Also, a DMAC digitising table is available for recording the location of all nodes from a mesh drawing using a scanning device on to punched cards. Element topology can also be recorded in this way. Certain three-dimensional mesh generation routines are available for simple-shaped blocks, which may be superimposed as much as possible to represent real structures.

5. ELEMENT TYPES AVAILABLE

The element types available in Phase I of BERSAFE are shown in Figures 1 to 4. In all, twenty different types are available, and are categorised under plane stress, plane strain, axisymmetric and three-dimensional.

The plane stress elements (which also are used for mathematical plane strain) are designated EP6, EP6R, EP4, EP8, EP12 and EP16 (Figure 1). EP6 is the constant stress triangle with degrees of freedom in cartesian directions, and EP6R is similar but with degrees of freedom in polar coordinates. EP4 is a line element used for reinforcing. EP8 is a quadrilateral with linear displacement functions, and EP12 and EP16 are respectively the triangle and quadrilateral with midside nodes, having quadratic displacement functions. The effect of mathematical plane strain is simply obtained by modifying the material properties and using any of these elements.

The engineering plane strain element available is EP6P (Figure 2), which is similar to EP6. This element type must be used on its own, and only one stressing case may be used.

The axisymmetric elements are designated EX6, EX2, EX4, EX8, EX12 and EX16 (Figure 3). EX6 is the axisymmetric version of EP6, EX2 and EX4 are used in reinforcing, being respectively a hoop element and an axisymmetric line element. The elements, EX8, EX12 and EX16 are the axisymmetric versions of EP8, EP12 and EP16 respectively.

The solid elements are designated EZ12, EZ24, EZ45, EZ45R, EZ60, EZ60R and EZ96 (Figure 4). EZ12 is a simply tetrahedron with constant stress. The elements EZ24, EZ60 and EZ96 are brick-shaped elements having respectively none, one and two nodes along each side. The element EZ45 is a triangular prism, and EZ60R and EZ45R are versions of EZ60 and EZ45 respectively with degrees of freedom in cylindrical coordinates.

In the figures, for all elements the nodes are numbered in the sequence for definition on the topology cards, the first node being any corner node. Various types of load may be applied to each element, including line loads, gravity loads, centrifugal loads and temperature loads. Edge or pressure loads may be directly applied to non-constant stress elements, otherwise the equivalent nodal loads have to be applied (the nodal allocation being easily calculated).

Except for engineering plane strain, any suitable mixtures of different element types may be made. The coordinate systems for specifying input values and calculating stresses are independent and do not depend on the type of element being used. The directions of prescribed displacements and decouplings depend on the individual element degrees of freedom. Elements with midside nodes may have linear or quadratic edge geometry, and EZ96 (with two
nodes on each side) may have linear, quadratic or cubic edge geometry.

6. **EXAMPLES OF APPLICATIONS USING BERSAFE**

A large number of nuclear applications have been analysed using the BERSAFE system. The three examples described below have been chosen as typical cases, each being three-dimensional and using the quadratic displacement elements of type EZ45, EZ45R, EZ60 and EZ60R. These particular elements have proved to be very effective and easy to use for arbitrary three-dimensional shapes.

6.1 **Strains in Prestressed Concrete Pressure Vessel**

A typical podded boiler prestressed concrete pressure vessel subjected to internal pressure has been analysed using three-dimensional finite elements of the type EZ60R and EZ45R (Carmichael [5]). Due to the presence of holes in the vessel, an axisymmetric mesh could not be used and so a sector, bounded by symmetric faces, was considered. Two meshes were used, one being much coarser than the other (Figures 5 and 6). The results from the two representations are very similar indicating that the simpler mesh was fine enough for reasonably accurate results. Experimental results have also been obtained and compare well with the computed results. Typical strain components from these comparisons are shown in Figure 7.

6.2 **Cylinder-Cylinder Intersections**

The stress analysis of cylinder-cylinder intersections is an important aspect of plant design. Consequently, several such geometries have been studied using three-dimensional meshes and EZ60 and EZ45 element types. Figure 8 shows a typical geometry. Early work was concerned with designing meshes which were sufficiently fine to give generally accurate results and yet not overrefined to exceed the available computer limits (Hellen and Money [6]). This work included the assessment of how many elements were required around a half cylinder for good accuracy, how many elements were required along the cylinder to accurately accommodate stresses due to circumferential loads, and how many elements were required through the thickness of the joined cylinders. Most of the studies to date have been concerned with internal pressure loading, and for this case, only one quarter of the structure need be analysed. Results from a similar geometry to Figure 8 are shown in Figure 9, where excellent agreement is obtained for the hoop and axial stress components when compared with experimental results.

Other types of loading considered include thermal strains. For this case, the asymmetry of the temperatures requires that a half of the total structure be analysed. Using the temperature transient system FLHE, the temperatures at the time of maximum temperature difference were calculated and used as thermal loads for a stress analysis using BERSAFE (Fullard [7]).

6.3 **On Thin Cylinders and Cooling Towers**

The excellent performance of the EZ60 type elements in the cylinder-cylinder problems, which are moderately thick shells, led to the investigation of their performance for thinner structures. If they behaved well down to very thin cases, structures containing thin shells joined to thicker bodies either abruptly or gradually would be readily possible, retaining complete compatibility. Such structures include thin-walled cylinder-cylinder intersections.
Initially, a simple pressurised cylinder was analysed with one end encastre, and with a radius-thickness ratio of 200:1. The refinement of elements near the wall was adjusted until the deformations and stresses accurately followed the curves from theoretical analyses, keeping the same number of elements in the circumferential direction. The element lengths away from the wall were expressed in terms of the parameters of the cylinder so that they could be used for any radius-thickness ratios. Similar lengths were derived for pin-fixed end restraints. These lengths were verified for ratios of 500:1 and 20:1 with equally good results.

To study the effect of slight radial variations, a cooling tower structure was analysed. In this case, the profile is hyperbolical. A coarse mesh with ten elements through the height and six around the circumference was used with an encastre restraint at the ground. Two loading cases were applied, one with uniform external pressure and the other with a true wind loading. The former gave very close agreement to a finite difference program, thereby verifying the derived element lengths from the ground. However, the rapidly varying wind loading case around the half-circumference was not accurately represented by the six circumferential elements. Therefore, both cases were repeated on a finer mesh (Figure 10) with twelve elements through the height and twelve elements around the circumference. In both meshes, automatic mesh generation was used and a special pressure routine in BERSAFE was used to calculate the equivalent nodal loads accurately for the wind loading case. Both these facilities greatly reduced the amount of input preparation. The wind loading case results for the finer mesh gave deflections which agree very closely to an alternative finite element system (ASKA) using compatible triangular plate elements, and a theoretical analysis due to Albasiny and Martin [8], at N.P.L. (Figures 11 and 12).

It is concluded that the performance of EZ60 type elements for certain very thin structures is very good, and enables those structures which also contain thicker components to be analysed with no loss of compatibility at the join of the two regions. For shell elements, the compatibility problem can be serious.

ACKNOWLEDGEMENT

This paper is published by permission of the Central Electricity Generating Board.
REFERENCES


Figure 1  Plane stress and mathematical plane strain elements

For the deformed shape (shown dotted) the $u$ displacement is permitted such that the total axial force, $R_x$, is zero.

Figure 2  Engineering plane strain element
Figure 3  Axisymmetric elements
Figure 4  Solid elements
Figure 5  Coarse mesh for concrete pressure vessel

Figure 6  Fine mesh for concrete pressure vessel
Figure 7  Measured and predicted displacements in concrete pressure vessel

Figure 8  Mesh for a typical cylinder-cylinder geometry
Figure 9  Hoop and axial stress components for a typical cylinder-cylinder geometry

Figure 10  Fine mesh for cooling tower
Figure 11  Horizontal deflection at top of cooling tower

Figure 12  Horizontal deflection at throat of cooling tower
R. W. CLOUGH, U. S. A.

I assume the results presented which were obtained by a two point Gauss integration procedure were based on an analysis procedure similar to that reported last January in IJNME by Zienkiewicz et al. He was using reduced order integration for all strain energy components. Our experience at the University of California Berkeley, is that a selective reduced order integration can be more reliable. We reduce the order integration primarily with reference to the shear strain energy. It would be interesting to see how well this technique would work with your example.

T. K. HELLEN, U. K.

The presented results were obtained using the isoparametric 20-node hexahedron elements in thin shell situations. It has been shown that thin element works well in a variety of thin shell situations with the standard integration rules. The use of reduced integrating rules on all strain energy components gives excellent results in problems where bending effects dominate and these are also the conclusions of Zienkiewicz et al. The example of a pressurised thin cylinder with an encastré end showed that, with reduced integration rules, good results occurred near the encastré end, where bending effects were considerable, however, in the membrane regions the results were poor. This suggests that it is too severe to reduce integration for direct stress components in membrane situations. It would be interesting to try this example with elements containing selective reduced integration rules.