Computers & Structures, Vol. 7, pp. 303–313. Pergamon Press 1977. Printed in Great Britain

A GENERAL ISOPARAMETRIC FINITE ELEMENT PROGRAM SDRC† SUPERB‡

V. T. NICOLAS and E. CITIPITIOGLU
Structural Dynamics Research Corporation, 5729 Dragon Way, Cincinnati, OH 45227, U.S.A.

(*Received* 15 *November* 1975)

Abstract—SDRC SUPERB is a general purpose finite element program that performs linear static, dynamic and steady state heat conduction analyses of structures made of isotropic and/or orthotropic elastic materials having temperature dependent properties. The finite element library of SUPERB contains isoparametric plane stress, plane strain, flat plate, curved shell, solid type curved shell and solid elements in addition to conventional beam and spring elements. Linear, quadratic and cubic interpolation functions are available for all isoparametric elements. Independent parameters such as displacements and temperatures are obtained from SUPERB using the stiffness method of analysis. The remaining dependent parameters, such as stresses and strains, are evaluated at element gauss points and extrapolated to nodal locations. Averaged values are given as final output. The graphic capabilities of SUPERB consists of geometry and distorted geometry plotting, and stress, strain and temperature contouring. Contours are plotted at user defined cutting planes for solids and at top, middle or bottom surfaces for plate and shell types of structures.

In the first part of this paper, the program capabilities of SUPERB are summarized. Extrapolation techniques used for determining dependent nodal parameters and for contour plotting are explained in the second part of the paper. Behavior of standard, wedge and transition type isoparametric elements and the effect of interpolation function orders on accuracy are discussed in the third part. The results of several illustrative problems are included.

INTRODUCTION

Commencing in the 1960s, the development of *isoparametric* finite elements was a significant advance in numerical methods of analyzing structures. Since isoparametric elements usually allow more precise structural discretization, they provide a decided advantage over conventional, straight-sided finite elements, because a smaller total number is normally required to model a structure. Furthermore, since displacements are interpolated with higher order functions, predictions of displacements and stresses with isoparametric elements are as accurate as results obtained from advanced conventional types of finite elements.

"Isoparametric" is a generic name given to finite elements whose physical boundaries are described by so called "shape functions" which interpolate node point coordinates over the domain of the element. The same shape functions are also used to interpolate nodal displacements. The shape functions which are used to interpolate the nodal coordinates and displacements are expressed in terms of dimensionless quantities which are referred to as "intrinsic" or "natural" coordinates. Equations for element stiffness are transformed from physical coordinates to natural coordinates by a conformal mapping procedure. In general, the resulting equations are extremely complex and numeric values for element stiffness and other parameters must be obtained by numerical integration methods.

The popularity of iosparametric finite elements has grown each year since their original introduction.This is due to the successful solution of problems in many areas, including plane and three dimensional elasticity, plates, general shells, heat transfer and fluid flow. The pioneering work on the stiffness formulation of these elements is credited to unpublished work by I. C. Taig in the early 1960s. The introduction of the element in the technical literature was by Irons[1] in 1966. Irons's paper contained references to plane and solid type elements, but numerical results were not presented. Other early literature[2, 3] was also concerned with plane and solid type elements. In 1968, Ahmad *et al.*[4] introduced axisymmetric isoparametric elements, and, later, in 1970, the same authors presented a general, isoparametric plate and curve shell element[5].

Early results with the isoparametric type shell elements indicated that they were excessively stiff in a thin shell situation. Subsequent work by Pawsey[6] and Zienkiewicz[7] in 1971 showed that the poor performance was caused by large amounts of spurious shearing strain induced by the assumed displacement interpolating functions when the element was undergoing simple bending deformations. These authors further showed that a more flexible element could be achieved by selective or reduced numerical integration methods. From an accuracy viewpoint, solutions obtained by using reduced numerical integration for isoparametric thin shell elements either surpassed or were at least competitive with similar solutions obtained using conventionally formulated thin shell finite elements.

Thus, by 1972, a significant number of different type isoparametric finite elements had appeared sporadically in the literature and had been shown to be capable of providing accurate solutions to a wide variety of structural problems. Although various finite element computer programs employed a limited number of isoparametric element types at this time, there was no general purpose, user-oriented, finite element program specifically written to take advantage of the full spectrum of these powerful new elements. It was at this time that Structural Dynamics Research Corporation, SDRC, embarked upon a new product development effort to produce

†SDRC is a service mark of Structural Dynamics Research Corporation.

‡Presented at the Second National Symposium on Computerized Structural Analysis and Design at the School of Engineering and Applied Science, George Washington University, Washington, D.C., 29–31 March 1976.

an efficient, state of the art, general purpose finite element program taking maximum advantage of isoparametric finite element technology. The result of that effort is a program called SDRC SUPERB (*SUPER BEST*) that has now superseded a previous SDRC finite element code named BEST[8] that employed conventional finite elements, exclusively. The SUPERB computer program has been in commercial use since April of 1974. Thousands of simple and complex structural models have been analyzed since that time. Many users have checked their results by load-deflection and/or strain gage tests and verified the accuracy of the isoparametric element behavior. The purpose of this paper is to present the basic capabilities of the SDRC SUPERB program, to describe some of the special methods and techniques that were employed in utilizing the isoparametric elements, and finally to present some conclusions and observations relative to the behavior of these elements in various structural models.

1. PROGRAM FEATURES

1.1 *Static structural analysis*

The static analysis capability of SDRC SUPERB (currently Version 4.0) is based on small strain, small displacement theory and assumes all materials are Hookean in behavior. The library of available structural elements consist of both conventional and isoparametric types that are listed in Table 1. The interpolating functions of all isoparametric elements are based on the "serendipity" family of shape functions as described in Ref. [9].

Table 1. SUPERB structural element library

Element	Type	Nodal Degrees of Freedom
1. Plane Stress (2-D)	Isoparametric	u, v
2. Plane Strain (2-D)	Isoparametric	u, v
3. Axisymmetric Solid (2-D)	Isoparametric	u, v
4. Middle Surface Flat Plate (2-D)	Isoparametric	w, θ_x, θ_y
5. Solid (3-D)	Isoparametric	u, v, w
6. Middle Surface Shell (3-D)	Isoparametric	$u, v, w, \theta_x, \theta_y, \theta_z$
7. Solid Type Shell (3-D)	Isoparametric	u, v, w
8. Straight Beam (3-D)	Conventional	$u, v, w, \theta_x, \theta_y, \theta_z$
9. General Stiffness or Flexibility Element (3-D)	Conventional	$u, v, w, \theta_x, \theta_y, \theta_z$
10. Linear Spring (3-D)	Conventional	u, v, w
11. Linear Spring (2-D)	Conventional	u, v

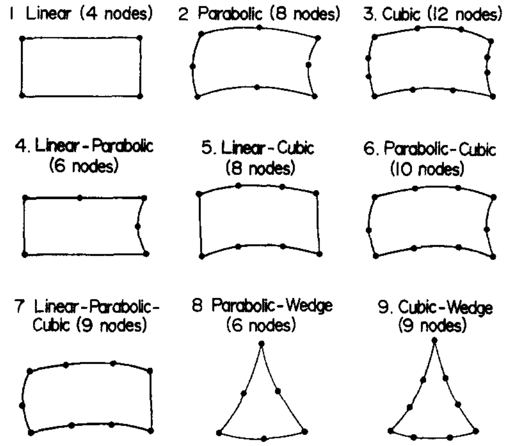


Fig. 1. Planar type isoparametric elements available in SUPERB.

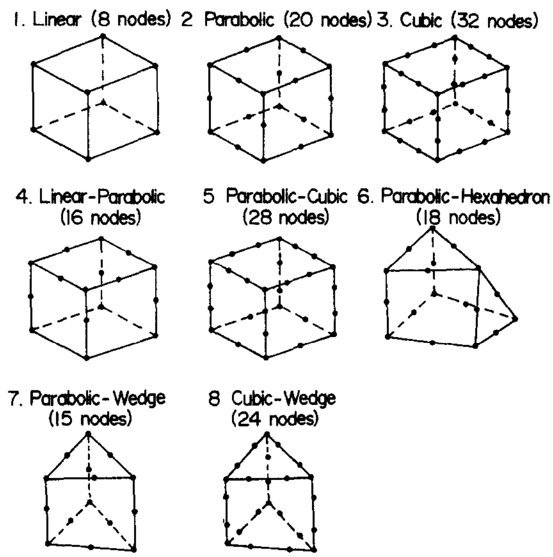


Fig. 2. Solid type isoparametric elements available in SUPERB.

Planar type elements. The plane stress, plane strain, axisymmetric solid and middle surface flat plate elements are planar type elements whose nodes must all be on a flat surface. Plane stress, plane strain and axisymmetric solid isoparametric elements are formulated using two dimensional elasticity equations. The middle surface flat plate element is based on Ahmad's formulation[5] with stiffness properties evaluated by reduced integration. Figure 1 shows the nodal patterns and shape function orders that are available for these types elements.

Solid elements. The solid type isoparametric elements are formulated using three dimensional strain-displacement elasticity equations. The available nodal patterns and shape function orders are shown in Fig. 2.

Middle surface shell. The middle surface shell element is based on Ahmad's formulation[5]. Element stiffness, however, is obtained by reduced integration as outlined in Ref. [6]. Element geometry is defined in three dimensional space by node points located on the middle surface of the element and nodal thickness values. Variable element thickness is obtained by interpolating specified nodal thickness values using the element shape functions. Each node on the shell possesses 3 translational and 3 rotational degrees of freedom that are compatible with nodal degrees of freedom on beam elements. Nodal patterns and shape functions available for the middle surface shell element are shown in Fig. 3.

Solid type shell. The solid type shell element is similar to Wilson's thick shell element used in the SAP program[10]. Geometry is defined by coordinates of node points situated on the top and bottom surfaces of the shell rather than on the middle surface. Even though the solid-type shell requires twice the number of nodal coordinates to define geometry as the middle-surface shell, users are not penalized since SUPERB only requires the node coordinates of either the top or bottom surface nodes and corresponding nodal thicknesses. Undefined node coordinates are generated automatically.

The standard element formulation exhibits excessive stiffness in thin shell situations similar to Ahmad's original formulation of the middle surface shell. Wilson[10] employed incompatible displacement modes to improve flexibility. The use of reduced integration in conjunction with the standard formulation improves flexibility, but still leaves the element 15–20% stiffer than a corresponding middle surface type shell element. Within SUPERB, the authors have modified the element formulation by introducing suitable changes in the stress–strain relation-

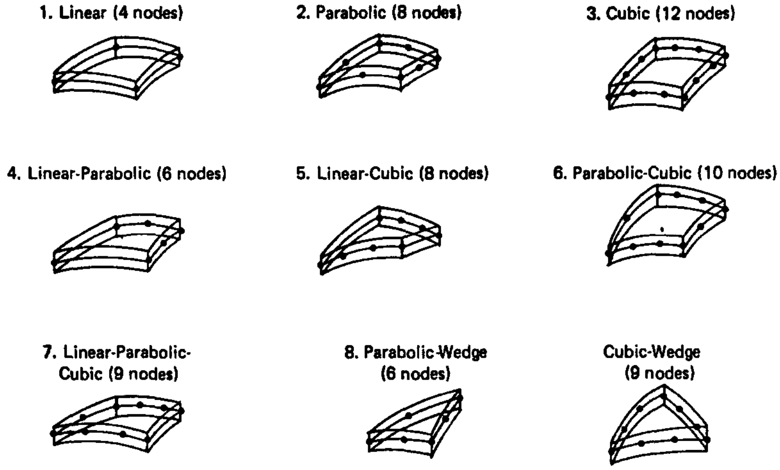


Fig. 3. Middle surface shell elements available in SUPERB.

ships which produce zero normal stress in the element thickness direction. This modification coupled with reduced integration gives results which are virtually the same as those obtained from the middle surface shell element.

Unlike the middle surface shell element, the nodal degrees of freedom on solid type shells are compatible with nodes on solid elements. However, a standard solid type shell element cannot be directly connected to a standard solid element, because abutting element faces would have incompatible node patterns and discontinuous displacements would occur at the interface. Since it is often desirable to connect solid type shell elements to solids, special elements were developed to make the transition. These elements provide displacement compatibility on their boundaries between either the standard solid element or the solid type shell element as illustrated in Fig. 4. The nodal patterns and shape function orders available for the solid type shell element are shown in Fig. 5.

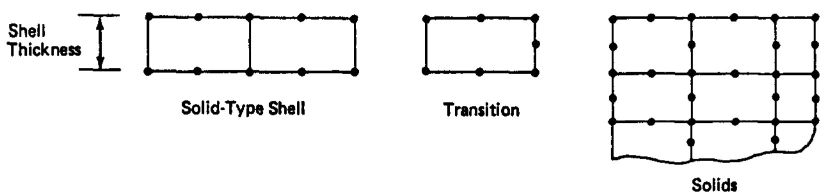
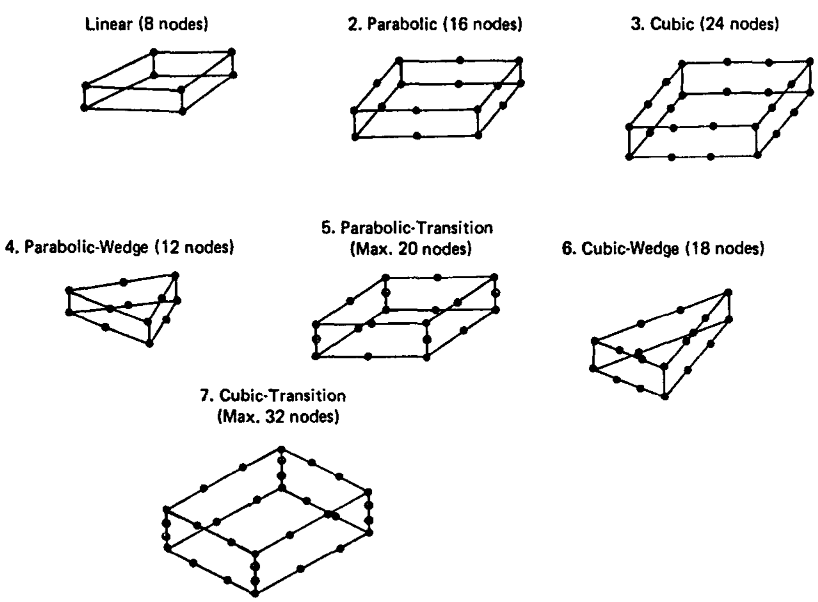


Fig. 4. Transition between solid-type shells and standard solid elements.



Note: Nodes indicated as • on the transition elements are optional and are defined as required by the user.

Fig. 5. Solid type shell elements available in SUPERB.

Loading. SUPERB will accept the following types of loads for static structural analysis runs:

1. Concentrated nodal forces applied in global or non-global directions at any structural node point.
2. Surface forces applied on any surface or edge of a finite element. Non-uniform distributions are permitted.
3. Body forces applied as constant linear accelerations in global coordinate directions or as a constant angular velocity about a global coordinate axis.
4. Thermal loads applied in terms of temperatures at each node point. Top and bottom surface temperatures can be specified at nodes of flat plate and middle surface shell elements to obtain thermal bending effects.
5. Non-zero displacement values in global or non-global directions at any node point.
6. Concentrated, uniform or linearly varying loads applied in global or member-oriented directions at any point between the end nodes of beam elements.

Output. Tabular data available at the conclusion of a static analysis run are:

1. Echo of input data.
2. Nodal displacements reported by ascending node number.
3. Stress values at node points and/or Gauss points of isoparametric finite elements. At each point where stresses are given, components of the stress tensor (directional values), principal stress values and the equivalent Von Mises stress value are reported. Stress reported at Gauss points are directly calculated using the assumed displacement functions and governing differential relationships. Stresses reported at node points are obtained by averaging the nodal stress values of all elements attached to the node. Element nodal stresses are obtained by interpolation of Gauss point stresses, rather than direct calculation. The method of interpolation is explained in the Section 2 of this paper. In the case of nodes at corner junctions of shell elements, where the structural surface is discontinuous and not smooth, averaged nodal stress values are invalid and are not reported. Instead, an appropriate message is printed in the list of nodal stresses and Gauss point stresses of the adjacent elements are automatically reported. In addition to a report listing nodal stresses in numerical node sequence, the user may also request resequencing and reporting of stresses in accordance with the magnitude of maximum principal, minimum principal or Von Mises stress values.
4. Strain values at node points and/or Gauss points of isoparametric elements. Strains are computed and reported in the same manner as stresses.
5. Directions of principal stress and strain axes.
6. Element nodal forces.
7. Reaction forces at locations of structural restraints.
8. Equilibrium balance at unrestrained nodal points.

1.2 *Steady state heat transfer*

SUPERB has the ability to perform finite element heat conduction analysis based on Poisson's equation for steady state temperature distribution. A full library of heat conduction elements which are compatible with SUPERB's structural elements has been implemented. These elements are listed in Table 2.

The nodal patterns and shape function orders for the isoparametric heat conduction elements are the same as those shown for the compatible structural elements in Figs. 1–3 and 5. At the conclusion of a heat transfer analysis, a nodal temperature file is created that can be directly entered as data in a subsequent structural analysis to obtain thermal stress results. Since heat conduction and structural elements share identical node patterns, the structural analysis run can be made with a few simple changes in the input data.

Table 2. SUPERB heat conduction element library

Element	Type	Compatible With Structural Element
1. Conducting Bar (1-D)	Conventional	Beam, Springs
2. Flat Conducting Surface (2-D)	Isoparametric	Plane Stress, Plane Strain
3. Axisymmetric Conducting	Isoparametric	Axisymmetric
4. Curved Conducting Surface (2-D)	Isoparametric	Middle Surface Plate Middle Surface Shell
5. Curved Conducting Shell	Isoparametric	Solid Type Shell
6. Solid Conducting Element (3-D)	Isoparametric	Solids

The program allows the input of temperature dependent thermal conductivity and average heat generation rates for each element. The final steady state temperature distribution is obtained by iteration when temperature dependent properties are given. The user may also permit heat exchange between element surfaces and the environment by specifying surface heat flux values and/or convective film coefficients.

Tabular output available at the user's option are:

1. Report of nodal temperatures.
2. Specified and/or convective heat flow from element surfaces.
3. Averaged heat flux values at node points.

1.3 *Dynamic analysis*

Dynamic analysis is currently performed in SUPERB by transmitting element stiffness and mass data to NASTRAN via a *N*ASTRAN *I*NTERFACE *P*ACKAGE (NIP) developed by SDRC[11]. The NIP series of programs was written to transmit dynamic characteristics of structural components, including stiffness and mass matrices, to NASTRAN to perform a dynamic analysis, and subsequently, to recover pertinent information from the NASTRAN analysis for final processing. Stiffness and consistent mass matrices for all structural elements are available to perform dynamic analysis. Both normal mode analysis and forced response analysis can be performed using SUPERB's isoparametric elements.

Dynamic analysis capability internal to SUPERB is under development and will augment the SUPERB-NASTRAN interface.

1.4 *Material properties*

Element material properties are permitted to be either isotropic or orthotropic. The orientation of orthotropic material axes is assumed to be constant within a given element, but can change orientation from element to element. The orientation of the orthotropic axes is defined with respect to the global axis for solid elements and with respect to local element oriented axes for the remaining element types. Nine constants which define the directional values of the modulus of elasticity (Exx, Eyy, Ezz), shear modulus (Gxx, Gyy, Gzz) and Poisson's ratio (γxx, γyy, γzz) are required to define the mechanical properties of an orthotropic material. Although these constants are independent, they are not arbitrary, because the stress–strain relationship must be positive definite in nature. That is, the material must store positive energy when it is deformed. This requirement is automatically verified by the program for all orthotropic materials before SUPERB execution is allowed.

Material properties are also permitted to have up to a 4th order polynominal variation with temperature. For static structural analysis runs, element stiffness coefficients are determined using material properties evaluated at the average element temperature. For heat transfer analysis, properties are also evaluated using average element temperatures and the steady state temperature distribution is obtained by iteration.

1.5 *Equation solver*

Generally, the solution of the linear load deflection equations constitutes the largest portion of the computer time spent on a finite element analysis. The method of solving these equations and the manner in which data are stored and retrieved during the solution have a profound influence on the solution time. SUPERB uses the wavefront method of Gaussian elimination[12]. This elegant technique, which eliminates a high percentage of operations on zeros during solution, is not affected by the node numbering scheme used on the structural model. Instead, the order in which variables are eliminated, and the size of the wavefront of active equations are controlled by the sequence in which the finite elements are processed during the solution phase. In SUPERB, elements are processed in the sequence in which they are defined in the input data rather than by preassigned element numbers. Reordering the sequence of elements to minimize the root mean square (RMS) wavefront (and solution cost) involves only simple changes in the input data. An element resequencing program is also available to automatically pre-process input data and produce a minimum or near-minimum wavefront of equations.

1.6 *Data input*

Typically, the preparation of input data, notably definition of element mesh, element connectivity and nodal coordinates, constitutes as much as 60–70% of the total cost of performing a finite element analysis. SUPERB offers many features to minimize the work of data preparation.

Coordinates of node points can be defined in standard Cartesian, cylindrical or spherical coordinate systems. Additionally, the user may define his own special Cartesian, cylindrical or spherical coordinate systems which can be offset and rotated with respect to global coordinate axes to simplify input of node coordinates. Also, extensive element and nodal coordinate generation schemes are available to assist in data generation. In particular, the coordinates of interior nodes on the edges of parabolic and cubic type isoparametric elements can be automatically generated from the coordinates of corner nodes. The interior coordinates are generated by linear interpolation in the same coordinate system used to define the corner nodes and, hence, the coordinates of interior nodes on cylindrical and spherical surfaces are accurately computed.

Besides the internal data generation capabilities in SUPERB, several external satellite programs have been developed to assist in data preparation. These programs are:

1. SDRC SUPER—a time sharing, interactive program for online preparation of SUPERB data files and generation of geometry and post plots. The program is run from low speed terminals operating at 10, 15, 30 or 120 characters per sec.
2. SDRC INTELLIGENT TERMINAL—a minicomputer based interactive system for finite element program data preparation. This system has local digitizing and graphic capability for entry and CRT display of element and node layout[18].
3. SDRC SUPERTAB—a capability similar to the intelligent terminal system except that it uses a time sharing computer and TEKTRONIX digitizing tablet and CRT for entry of digitized data.

1.7 *Pre-processing*

In a pre-processing mode, SUPERB will:

1. Make geometry plots of an entire finite element model or selected portions of the model. The edges of all elements within the viewed volume are shown in the plots. Optionally, node locations, node numbers, element number, material code and element type code can be shown for each element. Plots can be obtained on either CRT devices or X-Y incremental plotters.
2. Perform a wavefront check to determine the maximum and RMS wavefront for a given element sequence. The program also lists the computer core requirements and estimates time needed for an analysis run.
3. Perform a distortion check on all isoparametric type finite elements and list an individual distortion parameter value for each element. This parameter, whose basis is explained in Section 2, provides a relative assessment of individual element distortion that may be used to qualitatively estimate potential loss of element accuracy caused by user defined element geometry. The geometry of elements with unacceptably low distortion parameters can be modified prior to the final solution run.

1.8. *Post-processing*

SUPERB offers a comprehensive post-processing graphics capability. The following types of post plots can be obtained on either CRT devices or X-Y incremental plotters:

1. Distorted geometry.
2. X, Y and Z normal stress contours.
3. XY, XZ and YZ shear stress contours.
4. Maximum and minimum principal stress contours.
5. Maximum shear stress contours.
6. Von Mises equivalent stress contours.
7. Maximum and minimum principal strain contours.
8. Maximum shear strain contours.
9. Displacement contours.
10. Isothermal contours.
11. Heat flux contours.

On plate, shell and planar type elements, contour plots are superimposed on the top, bottom or middle surfaces of the elements. For solid type elements, the contour plots are superimposed on cut faces that result from passing user defined cutting planes through the structure.

2. SPECIAL FEATURES AND METHODS

2.1 *Distortion parameter*

The use of higher order displacement fields in the formulation of conventional finite elements generally leads to better element performance. The inclusion of higher order polynominal displacement fields in the formulation of isoparametric elements, however, does not automatically assure that the element will attain its optimum level of performance, because distortions in the mapping process can diminish accuracy. Strong distortions severely increase element stiffness, and worse, non-unique mapping can occur due to unreasonable distortion.

Element distortion is caused by excessive curvatures or irregular distribution of boundary node points on the physical element defined by the user. This may result from intentional attempts to use elements with heavily distorted boundaries in difficult areas of a finite element model or, more frequently, from inadvertent errors in input data. In either case, it is important to determine if elements exist with high or unreasonable distortion in a completed finite element model before an expensive analysis run is attempted.

There is no simple quantitive parameter available to measure the distortion of an element and exactly predict degradation of element accuracy with respect to its optimum capabilities. Elements which are unreasonably distorted and exhibit nonunique mapping can be located by using the theorem of preservation of regions under a conformal mapping transformation. Under a transformation from the dimensionless (ξ, η, ζ) domain to the physical (x, y, z) domain, this theorem requires that the functional determinant or Jacobian defined as

$$\frac{\partial(x, y, z)}{\partial(\xi, \eta, \zeta)} = \begin{vmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \zeta} \\ \frac{\partial y}{\partial \xi} & \frac{\partial y}{\partial \eta} & \frac{\partial y}{\partial \zeta} \\ \frac{\partial z}{\partial \xi} & \frac{\partial z}{\partial \eta} & \frac{\partial z}{\partial \zeta} \end{vmatrix} = |J|$$

does not vanish anywhere in the domain. It is a simple matter to determine if the Jacobian becomes zero by looking for sign changes during element formulation. However, a simple mathematical theorem does not exist to identfy elements that have strong but not unreasonable distortion. To aid in identifying heavily distorted elements, a relative distortion parameter has been devised and instituted in SUPERB.

The distortion parameter evaluated in SUPERB makes use of the fact that the Jacobian is a constant when the physical element is a parallelepiped with evenly spaced nodes, and the value of the Jacobian is alternately raised and lowered in different portions of the element as boundaries are distorted or nodes shifted to uneven spacing. The lowest Jacobian values tend toward zero as element distortion is increased, and become zero at the point of unreasonable distortion. For solid type elements, the element volume can be determined as:

$$VOL = \int_{-1}^{1} \int_{-1}^{1} \int_{-1}^{1} |J| \, d\xi \, d\eta \, d\zeta$$

when the dimensionless parameters, (ξ, η, ζ) range from -1.0 to $+1.0$ in the natural coordinate system. Since the Jacobian is constant for parallelepipeds with evenly spaced nodes, the ratio of the value of Jacobian to the volume of the element is also a constant equal to:

$$\frac{|J|}{VOL} = \frac{1}{8}.$$

Hence, when element distortion exists, the ratio of minimum Jacobian value to element volume, $\min |J|/VOL$, will always be less than the value of 1/8 obtained for a parallelepiped. Based on the above logic, a normalized relative distortion parameter (DP) can be defined for solid type elements as:

$$DP = 8\left(\frac{\min |J|}{VOL}\right).$$

The analogous distortion parameter for planar elements is:

$$DP = 4\left(\frac{\min |J|}{VOL}\right).$$

The distortion parameter will have a maximum value of 1.0 for a parallelepiped. When elements have curved surfaces, faces that are not parallelograms or nodes that are irregularly spaced, the distortion parameter value will be less that 1.0. Experience to date indicates that elements with a distortion parameter that is less than about 0.2 may produce undesirable effects. The effect of a single distorted element will be very local and may be unimportant depending on the location of the element in the model. A series of adjacent elements with low distortion parameters will definitely produce local inaccuracies and may affect the overall accuracy of the analysis. Of course, DP values less than or equal to 0.0 are completely unacceptable since non-unique mapping is indicated.

2.2 *Stress computation*

With the smaller number of higher order isoparametric elements that can be used in a structural model, it is important to compute and report stresses in a manner that is meaningful to the user. Computations of stress at element centroids, which is usually done for conventional elements, is not adequate for isoparametric elements, because stresses can have large variations between sparsely spaced elements. Furthermore, element centroids are normally not at critical stress locations and the user is often required to perform the necessary interpolations. The node points which are used to define element geometry (some of which are always on exterior surfaces of the structure) are usually preferred by the analyst as locations for computing stress values. Stresses at structural node points give an excellent overall picture of stress distribution to the user.

The presentation of nodal rather than centroidal stress values is not simply a choice between different locations for calculating stresses. First, stresses do not generally possess interelement continuity in finite element analysis, so that elements connected to the same node may yield different stress values. Secondly, while stresses can be computed at any point in the element, it is known that values obtained by direct computation at node points can be in considerable error. Stresses are predicted more accurately at the Gauss points used for numerical integration. In fact, for elements requiring reduced integration, the Gauss points are the only locations where certain types of stresses are predicted correctly[6, 13].

Hinton and Campbell [14] have pointed out the need for a rational and consistent procedure for determination of node point stresses. Their study indicated that local element stress smoothing followed by simple nodal averaging produced excellent agreement between theoretical and predicted nodal stress values. They defined local element stress smoothing as an exact least squares fit of a smoothed stress field to the element Gauss point stresses. The assumed smoothed stress field was defined by interpolating shape functions that were one order lower than the shape functions used to interpolate the element displacement field. The study was limited to flat plate and planar type parabolic, isoparametric elements and 2×2 reduced Gaussian numerical intergration. For these conditions, the stress smoothing procedure is equivalent to bilinear interpolation of Gauss point stresses in the element's natural coordinate system.

Nodal point stresses are obtained in SUPERB by local stress smoothing and subsequent nodal averaging. However, local stress smoothing is achieved by Lagrangian interpolation of Gauss point stresses in the natural coordinate system of the element. This is equivalent to bilinear interpolation for a 2×2 Gauss point pattern such as discussed in Ref. [14].

In the smoothing procedure, care must be taken to ensure that the Gauss point stresses are evaluated in consistent directions to prevent intermixing of normal and shearing types of stress during the interpolation process. Stresses in planar and solid type elements are evaluated in the global coordinate directions which allows straightforward interpolation. Stresses in curved shell elements should be interpolated on surfaces parallel to the middle surface of the element rather than in global directions to preserve basic stress characteristics. In SUPERB, interpolation is performed on curved shell elements using stresses whose directions are defined by local Gauss point coordinate systems as shown in Fig. 6.

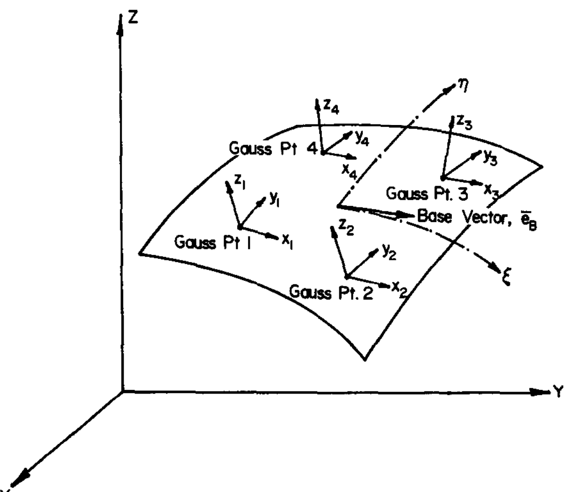


Fig. 6. Curved shell Gauss point coordinate systems.

A base vector $\bar{e}_B$ is established at the centroid of the element that is tangent to the middle surface of the shell in the local direction of the curvilinear ξ axes. At each Gauss point (i), the local Z_i axis is defined normal to the midsurface. The direction of the local Y_i axis is found by taking a vector cross-product between a vector parallel to the local Z_i axis and the base vector $\bar{e}_B$. The local X_i axis is determined by orthogonality requirements with respect to Y_i and Z_i. The resulting system of local Gauss point coordinate axes assures that stress components remain separated and are not intermixed during interpolation.

While local Gauss point coordinate systems are necessary for element stress smoothing in curved shells, they do not assure that the smoothed stresses at common nodes of adjoining elements are computed in the same direction. Hence, the smoothed stresses of each element must be transformed to a common set of axes before they can be averaged. A local coordinate system is therefore defined at each node for shell type element as shown in Fig. 7. The local Z_n axis is normal to the shell surface at the node point. The direction of the local X_n axis is found by the vector cross-product of the unit $\bar{j}$ vector in the global Y direction and a vector parallel to the local Z_n axis. Local Y_n is then determined by orthogonality requirements. In the special case when local Z_n is parallel to global Y, the local X_n axis is defined by a vector cross-product between the unit $\bar{i}$ vector parallel to the global X axis and a vector in the Z_n direction. For shell elements, Gauss point stress values are reported in the element Gauss point coordinate system and nodal stress values are reported in the local nodal coordinate system.

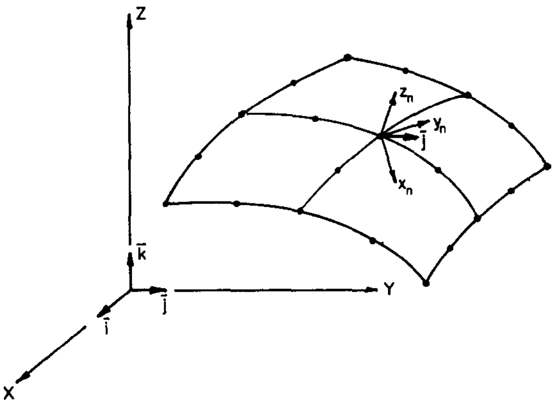


Fig. 7. Local nodal coordinates for shell elements.

2.3 *Contour plotting*

Stress smoothing and nodal averaging produces a rational pattern of discrete stress (strain) values at all structural nodes. It is only necessary to provide appropriate interpolation in order to plot contours.

One possibility would be to interpolate contour lines using the root stresses at element Gauss points and the averaged nodal stress values. A second possibility would be to use the original element shape functions to perform interpolation of stresses. Both methods have been tried on various problems and produce the same basic pattern of contour lines. The latter method which employs the element shape functions, however, tends to produce smoother contour lines and has been adopted for SUPERB post-plotting.

Figures 8 and 9 show contours of Von Mises equivalent stress for the cylindrical shell roof example given in Section 3. The contours in Fig. 8 were obtained by linear interpolation between the element Gauss point stresses and the averaged nodal stress values. The contours in Fig. 9 were obtained by using the averaged nodal stresses and element shape functions for interpolation. Both plots are essentially the same but the later contours are smoother.

3. ISOPARAMETRIC ELEMENT BEHAVIOR

In finite element applications, it is always desirable to use the minimum possible number of elements in a model in order to reduce the costs of data preparation and computation. On the other hand, convergence to "exact" solutions is achieved in finite element analysis when a structure is represented by an "infinite" number of elements. Thus, finite element application specialists must find a happy medium between the number of elements required to produce a "bare bones" description of structural geometry and that required to obtain acceptable

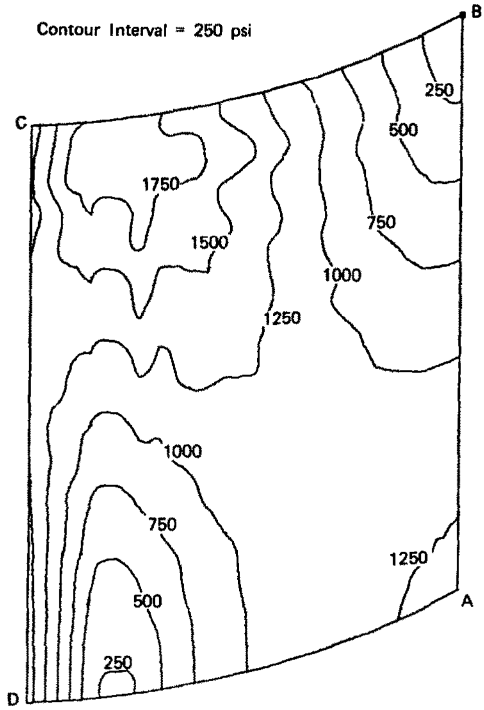


Fig. 8. Von Mises equivalent stress contours on cylindrical shell roof example obtained by linear interpolation between Gauss point and nodal stress values.

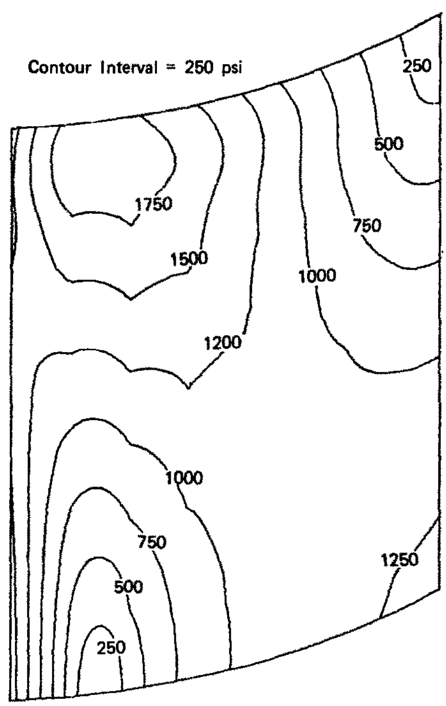


Fig. 9. Von Mises equivalent stress contours on cylindrical shell roof example obtained using element shape functions to interpolate nodal stress values.

accuracy from the finite element solution. A thorough understanding of and experience with the behavior of different finite element types under various loading conditions and structural configurations seem to be mandatory in order to make proper element selections in modeling structures. The Report [15] of the Task Committee of the American Society of Civil Engineers on Automated Analysis and Design is one of the large scale attempts in this area. Beyond the normal criteria necessary for successful finite element modeling, users of isoparametric elements must consider the additional effects of reduced integration and element distortion on the behavior of the elements. During the development and checking stages of the SUPERB program, more than six hundred problems were solved by the authors using elements from SUPERB's isoparametric element library. Results of four representative problems together with general conclusions on isoparametric element behavior are presented in this section.

3.1 *One element curved cantilever beam*

A one element curved cantilever beam of length, a, having central angles 0°, 90°, 60° and 30° subjected to an end moment, M, is shown in Fig. 10. The problem was

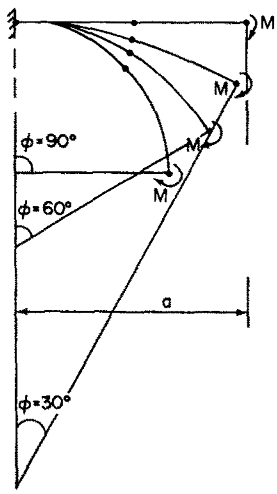


Fig. 10. One element curved cantilever beam [10].

originally presented by Wilson [10] and solved by the use of a parabolic thick shell element employing incompatible displacement modes to improve flexibility. The same problem was solved in SUPERB by using a parabolic Middle Surface Shell (MSS) element and a parabolic Solid Type Shell (STS) element. Element stiffnesses were formulated by reduced integration methods in SUPERB.

The ratios of end deflections obtained from the various finite element solutions to that of beam theory are tabulated for beam length to thickness ratios, a/t, of 5 and 50 in Table 3. SUPERB's shell elements exhibit exceptionally good performance in both the thick and thin shell domain over the entire range of curvatures. Wilson's thick shell formulation is excessively stiff for high curvatures in a thin shell situation.

Table 3. One element curved cantilever beam deflections

a/t	Element Type		Ratio of Maximum Deflections Δ Finite Element/Δ Beam Theory			
			$\phi = 0°$	$\phi = 30°$	$\phi = 60°$	$\phi = 90°$
5	SUPERB	STS	0.993	0.991	0.968	0.890
		MSS	0.993	0.987	0.986	1.008
	SAP – Thick Shell*		~1.0	0.98	0.95	0.85
50	SUPERB	STS	0.991	0.929	0.896	0.819
		MSS	0.964	0.902	0.906	0.935
	SAP – Thick Shell*		~1.0	0.93	0.27	0.15

*Values are scaled from Figure 5.9 in Ref. 10.

3.2 *Bending of circular bar by end force*

Figure 11 shows a circular bar (subjected to an end force P) which is modeled by 25 SUPERB parabolic plane stress elements. The problem was solved by Pulmano[16] using 25 special plane stress annular sector elements. Pulmano's plane stress element uses corner node displacements as well as displacement derivatives as degrees of freedom. End loads of magnitude $P/2$ were applied at nodes B and $B1$ for Pulmano's analysis and also for SUPERB Load-1. For SUPERB Load-2, the total load P was equally distributed among the nodes on edge B-$B1$.

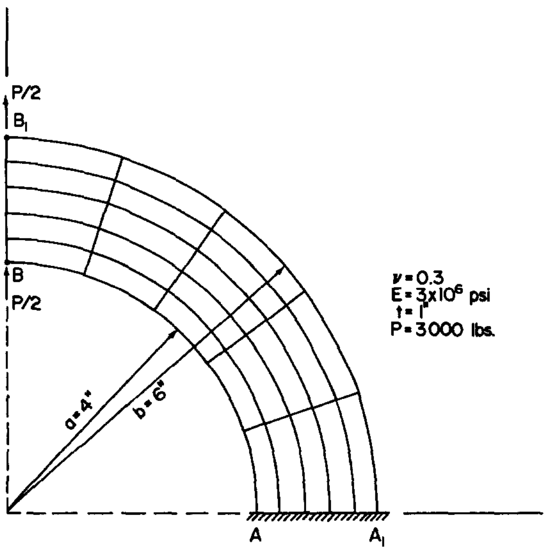


Fig. 11. Bending of circular bar by end forces [16].

End deflections and hoop stresses at the fixed end obtained by Pulmano and by using SUPERB plane stress elements together with a theory of elasticity solution are presented in Table 4. The results obtained by using SUPERB are in close agreement with those of the theory of elasticity. Hoop stresses calculated by SUPERB deviate from the theoretical results less than Pulmano's, indicating that the local element stress smoothing done in SUPERB is accurate.

3.3 *Cantilever cylindrical shell*

A quarter of a circular cylindrical shell is fixed along one edge and subjected to uniformly distributed edge load in the x-direction as shown in Fig. 12. The problem was solved by using Middle Surface Shell (MSS), Solid Type Shell (STS), regular Solid, Solid-to-STS transition, MSS wedge and STS wedge elements. Both parabolic and cubic displacement orders were used and reduced integration

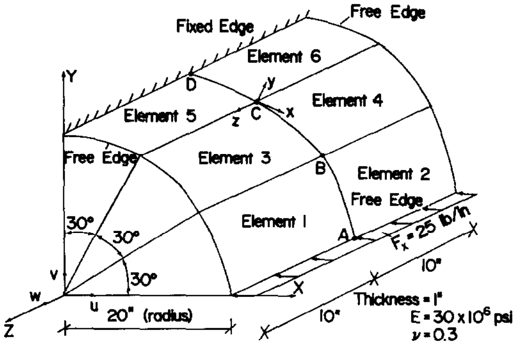


Fig. 12. Cantilever cylindrical shell subjected to a uniformly distributed edge load.

was employed in all cases. The mesh indicated in Fig. 12 was used for all element types except wedges. For wedges, each rectangular element shown in Fig. 12 was replaced by two wedges. When transition elements were used, elements 4 and 5 were modeled by regular solid elements. Mid-surface displacement and circumferential stress at nodes A, B, C and D on the top and bottom surfaces are presented in Table 5 together with beam theory results.

3.4 *Cylindrical shell roof*

A cylindrical shell roof subjected to 90 psf self weight is shown in Fig. 13. This example has been used for comparison purposes by many authors [6, 7, 9, 13], and the original solution is credited to Scordelis and Lo [17].(The authors, however, did not find the example in the mentioned Ref. [17].) To compare the performance and convergence of isoparametric elements with different order displacement fields and element mesh sizes, the example was solved by using 1×1, 2×2, 4×4, 8×8 meshes on one-quarter of the shell roof. SUPERB's STS and MSS elements with parabolic and cubic displacement orders were used. Selected displacements and nodal

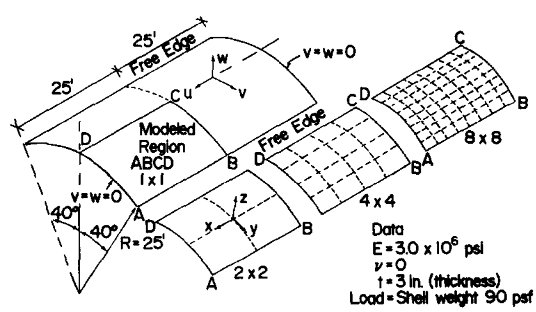


Fig. 13. Cylindrical shell roof, mesh configurations, displacement and stress reference axes.

Table 4. End deflections and hoop stresses of a circular bar under end force

r/a	End Deflection at B-B_1, in inches			Hoop Stresses, σ_θ in psi at A-A_1			
	Ref. 16	SUPERB Load-1	SUPERB Load-2	Ref. 16	SUPERB Load-1	SUPERB Load-2	Elasticity Solution
1.0	0.14946	0.15238	0.15015	27748	28501	28559	27664
1.1	0.14814	0.14992	0.14995	13705	14503	14464	14883
1.2	0.14765	0.14958	0.14984	4630	4425	4422	4541
1.3	0.14780	0.14971	0.14998	−3182	−4209	−4155	−4159
1.4	0.14848	0.15031	0.15029	−10557	−11609	−11688	−11714
1.5	0.14963	0.15213	0.15063	−19826	−20099	−20025	−18442

Exact Deflection (for r/a = 1.5) : 0.15064
Load - 1 : Load applied at B and B_1 (same as Ref. 16)
Load - 2 : Load equally distributed on edge B - B_1

Table 5. Displacements and stresses of cantilever cylindrical shell

Order of Displacement Function	Element Type	Mid-Surface Displacement (in inches) Node A −10u	Node A −10v	Node B −10u	Node B −10v	Node C −10u	Node C −10v	σ_x Circumferencial Stress (psi) Node A Top	Node A Bot.	Node B Top	Node B Bot.	Node C Top	Node C Bot.	Node D Top	Node D Bot.
Parabolic	MSS	0.580	0.369	0.226	0.277	0.033	0.091	2	22	1456	−1697	2532	−2939	3063	−3345
	STS	0.581	0.369	0.226	0.276	0.033	0.091	80	−58	1494	−1741	2552	−2958	2968	−3241
	Solid	0.561	0.350	0.216	0.260	0.030	0.082	−51	62	1579	−1821	2626	−3042	3155	−3465
	STS - Solid Transition	0.554	0.347	0.215	0.261	0.031	0.084	−40	68	1550	−1816	2608	−3053	3056	−3373
	MSS Wedge	0.487	0.314	0.193	0.237	0.030	0.081	−59	−247	1203	−1209	1658	−1997	2276	−3177
	STS Wedge	0.489	0.315	0.193	0.237	0.030	0.081	−2	−303	1212	−1217	1673	−1994	2233	−3100
Cubic	MSS	0.584	0.371	0.228	0.278	0.034	0.092	115	143	1535	−1654	2575	−2753	3274	−3147
	STS	0.584	0.371	0.228	0.278	0.034	0.092	116	144	1535	−1656	2555	−2732	3246	−3112
	Solid	0.562	0.350	0.217	0.260	0.030	0.082	109	164	1520	−1567	2498	−2759	3267	−3198
	STS - Solid Transition	0.555	0.350	0.215	0.261	0.031	0.086	139	140	1616	−1773	2742	−2824	3382	−3242
	MSS Wedge	0.445	0.283	0.175	0.212	0.026	0.071	−230	−263	1350	−753	1887	−2180	2305	−2413
	STS Wedge	0.445	0.283	0.175	0.212	0.026	0.071	−217	−276	1342	−746	1842	−2134	2265	−2373
Beam Theory*		0.572	0.364	0.224	0.273	0.033	0.091	0	0	1488	−1513	2676	−2620	2975	−3026

*EI is adjusted for $\nu = 0.3$

Table 6. Cylindrical shell roof displacements and stresses

Mesh Size	Order of Disp. Function	Type of Shell Element	Displacements (in inches) w_B	w_C	u_A	v_B	Averaged Nodal Stresses (psi) at Node B σ_X Top	σ_X Bot.	σ_{XY} Top	σ_{XY} Bot.	σ_Y Top	σ_Y Bot.	at Node C σ_X Top	σ_X Bot.	σ_{XY} Top	σ_{XY} Bot.	σ_Y Top	σ_Y Bot.	Relative Run Time
1 × 1		Ref.13	−3.808	0.746	0.094	−2.140	−58	1856	N.A.	N.A.	−37	−338	-	-	-	-	-	-	N.A.
	Parab.	MSS	−3.672	0.724	0.094	−2.057	−32	1820	−414	292	−110	−270	−446	−1341	−71	337	1592	−1408	1.0
		STS	−3.794	0.079	0.088	−2.139	−133	1858	−614	374	293	−382	−394	−1330	47	458	1370	−1438	1.0
	Cubic	MSS	−1.230	0.110	0.102	−0.516	419	1407	−383	−620	−348	−218	219	−403	−257	−133	1394	−737	1.53
		STS	−1.206	0.100	0.101	−0.504	402	1304	−464	−375	−222	−143	209	−370	−248	−137	1278	−835	1.67
2 × 2		Ref.13	−3.481	0.501	0.147	−1.841	1083	2020	N.A.	N.A.	−120	93	-	-	-	-	-	-	N.A.
	Parab.	MSS	−3.523	0.515	0.148	−1.865	1084	2026	−145	208	−156	120	126	−188	−221	−175	1416	−1619	1.8
		STS	−3.510	0.508	0.145	−1.861	1062	2084	18	64	−148	28	145	−194	−225	−173	1410	−1586	1.8
	Cubic	MSS	−3.129	0.431	0.137	−1.640	867	2910	−350	206	−589	−371	482	−571	−58	−3	1670	−1128	4.27
		STS	−3.219	0.450	0.138	−1.695	848	2994	−181	24	−611	−408	512	−574	−57	−4	1712	−1137	4.33
4 × 4	Parab.	MSS	−3.623	0.543	0.150	−1.911	1497	2366	−98	98	−66	75	58	−92	−20	−18	1328	−1519	5.33
		STS	−3.627	0.543	0.150	−1.913	1497	2370	−2	1	−58	67	58	−92	−20	−17	1325	−1517	5.47
	Cubic	MSS	−3.576	0.531	0.149	−1.883	1624	2540	−54	109	−94	−90	65	−162	−3	0	1289	−1435	16.80
		STS	−3.608	0.539	0.149	−1.902	1632	2656	28	28	−92	−96	70	−158	−3	−1	1305	−1440	17.33
8 × 8	Parab.	MSS	−3.619	0.543	0.149	−1.908	1623	2489	−24	24	−13	16	31	−103	−1	−1	1292	−1482	23.47
		STS	−3.617	0.542	0.149	−1.907	1624	2488	0	0	−12	15	30	−102	−1	−1	1290	−1430	24.00
	Cubic	MSS	−3.612	0.541	0.149	−1.904	1670	2533	−2	9	−8	−14	21	−110	0	0	1280	−1467	78.00
		STS	−3.617	0.542	0.148	−1.906	1672	2636	4	4	−7	−14	21	−109	0	0	1280	−1466	80.80
Classical Sol. (Ref. 13)			−3.703	0.526	0.161	−1.963	1519	2788	0	0	0	0	-	-	0	0	-	-	N.A.

stresses obtained from Ref. [13]) along with the results of the SUPERB analyses are given in Table 6. The last column of Table 6 contains the relative run times of SUPERB solutions on a CDC 6500 computer. Stress directions correspond with the local x, y, z axes shown in Fig. 13; displacements u, v and w are in the global X, Y, Z directions respectively.

3.5 *Conclusions*

The following conclusions relative to the behavior of isoparametric elements have been derived from the results of problems presented in this paper and many other problems solved by the authors.

1. Transition elements tend to be stiffer than their parent elements but do not produce excessive loss in accuracy, if they are properly used.

2. Wedge elements are significantly stiffer than the corresponding standard elements and their use in a model should be minimized.

3. Solid type shell and middle surface shell elements give approximately the same results when an appropriate reduced integration technique is used.

4. Shell like structures modeled with solid isoparametric elements are in the order of 10–15% stiffer than the same structure modeled with either solid shell or middle surface shell elements even when appropriate reduced integration methods are used.

5. In general, isoparametric elements employing parabolic shape functions to interpolate geometry and displacements seem to be the most cost effective.

6. Isoparametric elements employing linear shape functions require a large number of elements to obtain acceptable results. Their use should normally be avoided.

7. The cost of using elements with cubic shape functions is not in proportion to the increase in accuracy. Cubic elements are only justified under special circumstances.

4. SUMMARY AND CONCLUSIONS

SDRC SUPERB is a general purpose finite element program that primarily uses isoparametric finite elements. A program overview, special methods and techniques employed in utilizing the isoparametric elements and observations relative to the behavior of these elements have been presented.

Since 1974, SDRC SUPERB has been used to solve many practical stress analysis problems. The results of many of the analyses have been checked by experiments. From this experience, it can be concluded that isoparametric finite elements give cost effective and satisfactory results in practical stress analysis problems.

Acknowledgement—The authors acknowledge the helpful assistance of Dr. Russell F. Henke, Vice President of Operations, SRDC, in the presentation of this paper.

REFERENCES

1. B. M. Irons, Engineering applications of numerical integration in stiffness methods. *AIAA J.* 4(11), 2035–2037 (1966).
2. J. Ergatoudis, B. M. Irons and O. C. Zienkiewicz, Curved,

isoparametric "quadralateral" elements for finite element analysis. *Int. J. Solids Struct.* **4**, 31–42 (1968).
3. J. G. Ergatoudis, Isoparametric elements in two and three dimensional analysis. PhD. Thesis, Univ. of Wales, Swansea (1968).
4. S. Ahmad, B. M. Irons and O. C. Zienkiewicz, Curved thick shell and membrane elements with particular reference to axi-symmetric problems. *Proc. 2nd Conf. Matrix, Meth. Struct. Mech., Wright-Patterson A. F. Base* (1968).
5. S. Ahmad, B. M. Irons and O. C. Zienkiewicz, Analysis of thick and thin shell structures by curved elements. *Int. J. Num. Meth. Engng* **3**, 419–451 (1970).
6. S. F. Pawsey and R. W. Clough, Improved numerical integration of thick shell finite elements. *Int J. Numer. Meth. Engng* **3**, 575–586 (1971).
7. O. C. Zienkiewicz, R. L. Taylor and J. M. Too, Reduced integration techniques in general analysis of plates and shells. *Int. J. Numer. Meth. Engng* **3**, 275–290 (1971).
8. *Structural Dynamics Research Corporation*, User's manual for BEST computer program (1970).
9. O. C. Zienkiewicz, *Finite Element Method in Engineering Science*, 2nd Edn, Chap. 7. McGraw-Hill, New York.
10. E. L. Wilson, Solid sap, a static analysis program for three dimensional structures, Rep. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkley, California (1972).
11. W. A. McClelland and A. L. Klosterman, NASTRAN for dynamic analysis of vehicle systems. *SAE Paper* 740326, 95–107. *Proc. Int. Conf. on Vehicle Structural Dynamics, Detroit, Michigan* (1974).
12. B. M. Irons, A frontal solution program for finite element analysis. *Int. J. Numer. Meth. Engng* **2**, 5–32 (1970).
13. R. D. Cook, *Concepts and Applications of Finite Element Analysis*, 1st. Edn. Wiley, New York (1974).
14. E. Hinton and J. S. Campbell, Local and global smoothing of discontinuous finite element functions using a least squares method. *Int. J. Numer. Meth. Engng* **8**, 461–480 (1974).
15. Task committee on automated analysis and design, guidelines for finite element idealization, *ASCE National Structural Engineering Convention*, Meeting Reprint 2504, New Orleans (1975).
16. V. A. Pulmano, Tapered plane stress annular sector finite element. *J. Engng Mech. ASCE* **101** (EM4), 349–360 (1975).
17. A. C. Scordelis and K. S. Lo, Computer analysis of cylindrical shells. *J. Am. Concr. Inst.* **61**, 539–561 (1964).
18. J. G. Kormos, Increasing productivity of machinery engineers with computed aided design, Americal Society of Agricultural Engineers, *Palmer House*, Chicago, Ill. (Dec. 1975).