A general numerical formulation for elastic contact of solids
Elangovan, P.T.

Published: 01/01/1978

Citation for published version (APA):
A general numerical formulation for elastic contact of solids

by

P.T. Elangovan

Groep Technische Mechanica Afdeling Werktuigbouwkunde
Technische Hogeschool, Eindhoven.
1977.

WE 78.05
Acknowledgement

I am highly grateful to professor J.D. Janssen, dr. ir. F.E. Veldpaus, ir. W.A.M. Brekelmans and ir. J.S.H.M. Wismans for their valuable discussions in formulating this work presented in this report. My gratitudes are also due to the staff in the computing centre for allowing to have as much computer time as I required during the course of this work. I am highly thankful to this university for offering me a fellowship, but for which I could not have carried out this work.

I acknowledge my highest admiration of this university and its facilities to carry out research works in Engineering and applied sciences and of faculty and students for their politeness and kindness during my stay here.
ABSTRACT

This report deals with a formulation to analyse a contact problem of two elastic solids. Being a problem of technologically importance, contact problem has been an attractive analysis for applied mathematicians and engineers. This problem has been analysed, for its simplest and restrained applications, with the available analytical tools of potential theories and theories applied to semi-infinite medium. With the advent of computer in numerical applications in scientific investigations, numerical formulations have been employed to this problem. One such numerical formulation is illustrated and a numerical example is included in this report.

This numerical formulation has employed a finite element displacement analysis and a modified Wolfe's linear programming technique for the analysis of a general contact problem without friction in the interface between solids. The finite element analysis is employed to calculate the stiffness matrix; this stiffness matrix is inverted, after separating the rigid body motions from the solid, to get the flexibility matrix relating the applied forces and elastic displacements in the solid at the probable region of contact. Since a contact problem is always indeterminate one, having more unknowns than the available equations in the formulation, optimization technique is employed to solve this problem. Initially, this problem is considered as a quadratic constrained minimization problem and later, it is reduced to a linear programming problem and Wolfe's modified program is employed. Due to the special nature of this problem, a unique solution is possible. A numerical example of the Herz's sphere-to-sphere contact is analysed. This formulation can be employed to a contact analysis of two solids of arbitrary shapes and multiply connected contact regions, as it is seen in the knee-joints.
CONTENTS

Acknowledgement ............................................. I
Abstract ....................................................... II
Contents ........................................................ III

A GENERAL NUMERICAL FORMULATION FOR ELASTIC CONTACT OF SOLIDS.

I. Introduction .................................................. I
II. Numerical formulations ...................................... I
III. A formulation for elastic contact of solids ............. I
IV. Numerical examples .......................................... I
    1. Mathematical programming technique ..................... I
    2. Contact analysis by the "Marc" F.E. System ............. I
    3. General formulation ...................................... I
V. Reference ...................................................... I
VI. Appendix. A. Separation of rigid body motions for
    elastic analyses .......................................... I
VII. Appendix. B. Isoparametric three dimensional 20 node
    hexahedron element ....................................... I

Computer programs ............................................. I
A GENERAL NUMERICAL FORMULATION FOR ELASTIC CONTACT OF SOLIDS.

1. INTRODUCTION

When two elastic bodies are brought in contact and pressed together by a forcing system, the two bodies deform and form a contact area. An analysis of a contact problem must be able to provide the following two particulars:

i) area of contact between the two bodies

ii) magnitudes and directions of the interfacial pressures on the contact area.

From these two details, it is possible to evaluate the deformations and stresses in the interface and in the interior of the solids. This problem is an indeterminate one since both the kinematic boundary conditions and the nature of the boundary forces are unknown.

The first attempt to solve this technologically important problem is by Hertz (1881); the well known analytical methods employed for semi-infinite medium are applied in the Hertz's theory to a certain limited class of problems in the contact mechanics. The Hertz's theory is briefly outlined here since this theory has been widely used with or without modifications and extensions. The following assumptions of Hertz simplify the contact problem.

i) contacting bodies are isotropic and linearly elastic

ii) linear dimensions of the contact area are small in comparison with the radii of curvatures defining the solids

iii) the solids are having smooth, frictionless and continuous surfaces in the contacting region.

By the first assumption, available linear elasticity theories can be employed to this problem. The second assumption leads to the simplifying approximation that the contact area is flat enough such that the well-known analytical methods for a semi-infinite solid and for a solid bounded by a plane can be employed. Also, the region of contact can be approximated by a second degree polynomial surface.
The third assumption leads to the limitation that only normal forces are acting on the contacting surface without tangential frictional or shearing forces.

The application of the Hertz's theory for elastic solids characterised by a general quadratic surface at the region of contact is illustrated by Timoshenko (1951). Hertz shows that for a general case i) the contact area is elliptic with axes $a, b$ depending upon the geometric properties like the principal radii and orientation of the planes of principal curvatures, ii) the magnitudes of these elliptic axes $a, b$ change in proportion to $P^\frac{1}{3}$ where $P$ is the applied normal load iii) the rigid body approach of points far away from the region of contact is proportion to $O^\frac{2}{3}$ and iv) the maximum pressure $p_o$ occurs at the centre of the ellipse with a magnitude of 1.5 times the average contact pressure.

The stresses arising in the Hertz contact problems are evaluated later by a Russian Researcher, Balyayev and detailed in the reference book of Timoshenko (1951). Because of the practical importance, Hertz's theory has been extensively tested by researchers and this theory agrees well as long as the assumptions hold good in test problems. It is found that this theory predicts accurately the shearing stresses in elliptic contact problems as long as the semi-major axes of the elliptic surface is as large as the minor axes of either of the two elastic solids.

Beyond the linear elastic limit, some of the relationships hold good but accuracy decreases as the divergence from the elastic theory increases. This theory has been extensively employed even now for many engineering field problems.

Hertz's theory has been extended by many researchers and the extensions are reviewed by Lubkin (1962) and problems arising in contact mechanics are classified and extensions to the Hertz's theory have been outlined by Kalkar (1975). The Hertz's theory has been extended to solids whose contacting surfaces are other than the general quadratic surface as assumed in the theory; also additional system of forces is introduced apart from the normal contact forces to take care of the tangential forces in moving or rolling contacts or the shear and frictional stresses in solids with rough surfaces.
Even in a simple Hertz problem, shear stresses may develop when the two solids have different elastic moduli; due to differential displacements, shear develops unless the surfaces are having zero friction which is almost impossible to achieve in practice. Extensions to the Hertz's theory are evaluated analytically by many researchers but with the advent of computer, numerical analyses have been also employed to analyse non-Hertzian problems. Since this report envisages a numerical formulation to analyse a general contact case, some of the numerical formulations employed in the contact mechanics are reviewed in the following section.

II NUMERICAL FORMULATIONS

The numerical methods employed to this class of problems are grouped in two categories.

1) the methods employing the Boussinesq's solution of a normal load on a semi-infinite medium with a numerical method of analysis or a mathematical programming technique.
2) finite element methods of analysis.

An interesting numerical method which employs mathematical programming procedure to this class of problems is developed by Conry and Seireg (1971) and Kalkar and Van Randen (1972). In this method, the flexibility matrix connecting discrete contact forces and the displacements, is evaluated by employing the Boussinesq's solution for a normal concentrated load on an elastic half space. This method is quite advantageous since it is not involved with iterations or incremental step loadings, but employs a well established programming procedure and in fact, this procedure does not go into the iterative steps of finding the best possible feasible solution but finds only once the unique solution available for this class of problems. It is possible to apply to any problem where contact area can be considerable to the overall dimensions of the contacting solids. Also, it is not limited only to the problems with quadratic surfaces at the contact region.
However, in this method only discrete forces representing small domains are evaluated finally, the contact area and stresses are less accurate depending upon the degree of closeness of the mesh points in the contact region. The other disadvantage of this method is that it is limited to a class of problems where half space theory can be employed; the problems are required of having at least two planes of symmetry in the region of contact and it is not applicable to asymmetric problems. This method is later extended to layered medium by Johns and Leissa (1974) incorporating the mathematical programming procedure and the Henkel transformation methods, which find the influence coefficients between the contact stresses and displacements in the multilayered media.

Another numerical method based on elastic half space theory is developed by Singh and Paul (1974) as a general method to solve problems of frictionless non-Hertzian type when surfaces at the contact region are not quadratic. The method employs the Boussinesq's solution and a family of closed curves to define the contact boundary. The advantage of this method is that the contact area can be accurately predicted unlike the other numerical methods but the method has only a limited application to normal contact problems.

For a simple case of beam on elastic foundation, a numerical method by the use of beam theory and Boussinesq's solution is developed by Conway and Farnham (1970). This method is employed to a centrally loaded beam on an elastic foundation with a bonded or unbonded contact. Depending upon the relative elastic moduli of the beam and elastic half space, the pressure distribution across the beam width is assumed as variable or constant. The contact area is divided along the length into finite number of rectangular sections and deflections are calculated at the centers of these rectangular sections. When contact length of the beam is unknown, the problem is solved by incrementing the contact length and arriving finally a converging solution. Eventhough this method is simple, it is employed for a limited application and here again incremental technique is employed for unknown contact area.
Finite element method has been used to solve the contact problem by incrementing the load or the normal displacement and checking for the new contact area at every incremental step. (Engel and Conway (1971), Back Burdekin and Conway (1973), MARC System (1974). The incremental formulation is quite time consuming even for a simple case. By the application of finite element and with the concept of differential displacements between two sets of nodal points, problems with known contact area are solved by Wilson and Parsons (1970). In this method, differential displacements are assumed between the points in the regions of contact in both solids. Stiffness matrices for both solids are evaluated and assembled as a single global matrix and differential displacements are applied thus removing the indeterminacy involved in these problems. However, the method is applied to problems of known contact area and axisymmetric situations only.

A very general finite element formulation is developed by Chan and Tuba (1971) to analyse problems with any shape and loading. Frictional forces are also considered in addition to normal pressures on the contact surface. This formulation is based on the displacement stiffness method; iterative process is involved to arrive at a converging solution. A notable advantage of the formulation is that the points on the contact region of one body are not connected to those of other body throughout the iterative cycles. The disadvantage of this method is that it involves an iterative relaxation procedure for a solution and loading is incremented up to the final applied load by a finite number of steps. Another finite element formulation for contact problem is developed by Francavilla and Zienkiewicz (1975) and it can be employed to a general contact problem. The formulation is concerned with only the points in the possible region of contact and the reduced stiffness matrix for those points are only handled. However, iterative process is required for problems of unknown contact area. At every iteration, the solution is checked for no tensile pressure in the contact zone and no interpenetration outside the assumed contact zone.

From this brief review of the numerical methods, it can be seen that it is quite relevant to formulate a contact formulation which should have wide and versatile applications with less computer time requirement and without time consuming iterative or incremental procedures.
In this report, an attempt is made to give a formulation based on i) the finite element method to arrive at the compliance coefficients for points at the possible region of contact of arbitrary shaped bodies and ii) the mathematical programming procedure to obtain the interfacial discrete contact forces and eventually the contact area in a straightforward manner. The formulation is illustrated in the following section.

This report includes some numerical examples based on three numerical methods, i) mathematical programming procedure in combination with the Boussinesq's solution of a normal load on an elastic half space as formulated by Conry and Seireg (1971) ii) standard incremental contact formulation by MARC programming system by the introduction of contact elements at the interface between contacting solids and iii) the present new formulation.
III A FORMULATION FOR ELASTIC CONTACT OF SOLIDS.

In an elastic medium, the elastic displacement at a point \( i \) (without the contribution of rigid body motions) due to discrete forces \( F_j \) acting points \( j \) (\( j = 1, 2, \ldots, N \)) is related as

\[
U_i = \sum_{j=1}^{N} a_{ij} F_j
\]

(1)

where \( a_{ij} \) are the flexibility coefficients for the point \( i \), due to the forces at \( j \), \( F_j \).

In the case of finite element displacement analysis, the continuum is represented by discrete nodal forces. The general relationship is

\[
[K] \{u\} = \{F\}
\]

(2)

where \([K]\) is the matrix of stiffness coefficients. Generally rigid body motions are removed with prescribed boundary displacement conditions and in case of no such conditions, they are prescribed at some points arbitrarily in the body to represent fully the rigid body motions of that particular body. A general approach is illustrated in the Appendix A.

When a particular region \( \Omega \), is of interest in the whole continuum, the Eq. 2 can be partitioned as follows

\[
\begin{bmatrix}
K_{NN} & K_{NC} \\
K_{CN} & K_{CC}
\end{bmatrix}
\begin{bmatrix}
U_N \\
U_C
\end{bmatrix}
= \begin{bmatrix}
F_N \\
F_C
\end{bmatrix}
\]

(3)

Then the above Eq. 3 is reduced in terms of the quantities at the points in the region \( \Omega \) as follows

\[
\begin{bmatrix}
K_{CC}
\end{bmatrix}
\{U_C\} = \{F_C\} - \{\bar{F}_C\}
\]

(4)
where

$$\begin{bmatrix} \bar{K}_{CC} \end{bmatrix} = \begin{bmatrix} K_{CC} \end{bmatrix} - \begin{bmatrix} K_{CN} \end{bmatrix} \begin{bmatrix} K_{NN} \end{bmatrix}^{-1} \begin{bmatrix} K_{NC} \end{bmatrix}$$

(5a)

$$\{ \bar{F}_C \} = \begin{bmatrix} K_{CN} \end{bmatrix} \begin{bmatrix} K_{NN} \end{bmatrix}^{-1} \{ F_N \}$$

(5b)

from the Eq. (4), we can write

$$\{ U_C \} = \begin{bmatrix} \bar{K}_{CC} \end{bmatrix}^{-1} \left( \{ F_C \} - \{ \bar{F}_C \} \right)$$

(6a)

or

$$U_i = \sum_{j=1}^{N} a_{ij} (F_j - \bar{F}_j)$$

(6b)

The Eq. (6b) is similar to the Eq. (1)

When an analysis is carried out with stiffness coefficients with reference to a global coordinates which are different from the necessary directions at the contact points locally (normal direction and tangential directions), the stiffness matrix is transformed as follows. Employing Eq. (4), we get

$$\left[ T \right]^T \begin{bmatrix} \bar{K}_{CC} \end{bmatrix} \left[ T \right] \{ U_C \}_C = \left[ T \right]^T \left( \{ F_C \}_C - \{ \bar{F}_C \}_C \right)$$

(7a)

$$\begin{bmatrix} \bar{K}_{CC} \end{bmatrix}_C \{ U_C \}_C = \{ F_C \}_C - \{ \bar{F}_C \}_C$$

(7b)

$$\{ U_C \}_C = \left[ T \right] \{ U_C \}$$

(7c)

where $[T]$ is the transformation matrix between the global coordinates at the contact points. The origin of the global coordinates is determined in the body 1, as in fig. (1a), according to the assumed artificial support conditions to keep the rigid body motions equal to zero and also it is the point where rigid body motions are applied to bring out the contact.
When friction is neglected at the contact surface, it is possible that displacements and forces acting parallel to the local normals of contact points are separated by matrix condensation as shown in Eq. (3) and (4). Flexibility coefficients are obtained as in the Eq. (6). The subscript $C$ is dropped in Eq. (7) for the later part of the formulation.

For any pair of contacting nodes $i$ in the region $\Omega$, deformation compatibility of two solids in contact to satisfy no interpenetration in any particular direction is stated as follows:

\[
\begin{align*}
U_{1,2} + U_{i,1} + \delta_i \geq \alpha_R & \quad (8a) \\
U_{i,1} + U_{i,2} + \delta_i - \alpha_R \geq 0 & \quad (8b)
\end{align*}
\]

where the subscripts $1,2$ denote the bodies in contact; compressive interfacial pressure in the contact area is assumed to be positive and corresponding resulting displacements $U_{1,1}$ and $U_{1,2}$ are positive; $\delta_i$ is the initial separation of the solids at $i$ and $\alpha_R$ is the rigid body displacement at $i$ of the body $1$ and is measured along the concerned direction as shown in fig. 1a. both $\delta_i$ and $\alpha_R$ are always positive ($\geq 0$).

When rigid body motions $\alpha_1, \alpha_2, \alpha_3 \ldots \alpha_m$ are applied at the origin of the global coordinates, the value of $\alpha_R$ is calculated as follows:

\[
\alpha_R = \left[ r_1, r_2, \ldots, r_m \right] \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \vdots \\ \alpha_m \end{bmatrix}
\]
The value of $m$ is 3 or 6 depending upon the analysis whether it is 2- or 3 dimensional one. $r_1, r_2, \ldots, r_m$ is a row matrix transforming the rigid body motions at the origin of the global coordinates to the required direction at $i$ on the contact surface; when friction is excluded in the contact analysis, $\alpha_R$ is evaluated along the local surface normal at $i$.

Substituting Eq. (6b) and (9) in Eq. (8b) we get:

$$\sum_{j=1}^{N} (a_{ij,1} + a_{ij,2}) F_j + \delta_i - \sum_{i=1}^{3 or 6} r_i a_i - \sum_{j=1}^{N} (a_{ij,1} F_{j,1} + a_{ij,2} F_{j,2}) \geq 0$$

(10)

Considering all points in the contact surface, we get

$$[S] \{F\} + \{\Delta\} - [R] \{a\} + \{C\} \geq 0$$

(11)

where $\{C\}$ is a vector of constant quantities for a given loading and support conditions of the solids and a vector element $c_{ij}$ is equal of the last term Eq. (10); $[S]$ is the total flexibility matrix of the two solids at the contact surface points and a component, $S_{ij} = a_{ij,1} + a_{ij,2}$; $[R]$ is the transformation matrix of the rigid body motions of the origin of the coordinates of the body 1 to the points on the possible surface of contact; $\{\Delta\}$ is the vector of initial separations between the pairs of contact points in the bodies 1 and 2 along their local surface normals.

The arbitrary support conditions are introduced in the body 1, as shown in fig.1b, to limit the rigid body displacements (or rotations) to zero and evaluate the elastic part of displacements initially; with these support conditions, it is possible to generate a rectangular cartesian coordinates at whose origin, rigid motions are employed to make a contact with the body 2; the transformation matrix of those rigid body motions at other parts of the body 1 is evaluated with these coordinates. However, in contact situation, force equilibrium is maintained only between the contact interfacial pressure (in this case, discrete contact forces) and the applies forces but not shared by those arbitrary supports. When an equilibrating body is subjected to rigid body motions at the origin, it is well known that the equilibrium condition is obtained from the condition that rigid body motions do not
contribute additional potential energy.

\[
([Ra] \{a\})^T \{F_a\} + ([Rc] \{a\})^T \{F_c\} = 0 \quad (12a)
\]

or

\[
[Ra]^T \{F_a\} + [Rc]^T \{F_c\} = 0 \quad (12b)
\]

where \([Ra]\) is the transformation matrix of \(\{a\}\) at 0 with respect to the directions and points of action of the applied forces and \([Rc]\) correspondingly for the supporting forces. In the case of a contact situation, \([Rc] = [R]\) as in Eq 11 and the equilibrium condition to be satisfied is

\[
[Ra]^T \{F_i\}_{\text{applied}} + [R]^T \{F\} = 0 \quad (12c)
\]

OPTIMIZATION TECHNIQUE

It is seen in eq(11), that the number of unknowns is more than the number of equations; hence, a mathematical programming technique is employed in the contact formalation to solve for the unknowns. To start with, a quadratic optimization problem is stated as follows:

Minimize

\[
\{ F \}^T ( [S] \{F\} + \{\Delta\} - [R] \{a\} + \{C\} ) \quad (13)
\]

with the following constraints

\[
[S]\{F\} + \{\Delta\} - [R] \{a\} + \{C\} \geq 0
\]

\[
F_i \geq 0 \quad [R] \{a\} \geq 0
\]

\[
[R]^T \{F\} + [Ra]^T \{F_i\} = 0
\]

Then the solution solves the following case (Cattle, 1966 and Kartanek and Jeroslow, 1967)

\[
\{F\}^T ( [S] \{F\} + \{\Delta\} - [T] \{x\} + \{C\} ) = 0 \quad (15)
\]

which is, in fact, the condition when contact is established. Conained nonlinear optimization technique can be employed to this problem. But, the modified simplex type lineair programming by Wolfe (1959) is employed to minimize this quadratic function with lineair contraints because
of its simplicity in programming. The programming technique is as follows.

Introducing a new set of non-negative variables \( \{y\} \) with the condition \( \{F\}^T \{y\} = 0 \), the quadratic minimization problem is reduced to the following linear programming formulation.

\[
\begin{align*}
[S \{F\} + \{\Lambda\} - [R] \{\alpha\} + \{G\} - [I] \{y\} &= \{0\} \\
\{F\}^T \{y\} &= 0 \ , \ F_i \geq 0 \ ; \ y_i \geq 0 \ ; \ [R] \{\alpha\} \geq \{0\} \\
[R]^T \{F\} + [R]^T \{F\} &= 0 \\
[R]^T \{F\} + [R]^T \{F\} &= 0 \\
[I] - \text{a unit matrix}.
\end{align*}
\]

If \( \{y\} \) is interpreted as a vector of separation between the pairs of points parallel to their respective local surface normals in the contact region, the new condition \( \{F\}^T \{y\} = 0 \) is a necessary condition in terms of interpretation of a contact problem.

a. When there is contact between a pair of opposing points

\( F_i > 0 \quad y_i = 0 \quad (17a) \)

b. When there is no contact

\( F_i = 0 \quad y_i > 0 \quad (17b) \)

The modified simplex type Wolfe's algorithm is employed as follows.

The function to be minimized is the artificial object function

\[
f(z) = \sum_{i=1}^{n+m} z_i \quad (18a)
\]

subject to the following constraints

\[
\begin{align*}
-[S \{F\} + [R] \{\alpha\} + [I] \{y\} + [I] \{z_1\}] &= \{\Lambda\} + \{c\} \\
-[R]^T \{F\} + \{z_2\} &= [Ra]^T \{f_1\} \\
F_i \geq 0 \ ; \ y_i \geq 0 \ ; \ z_i \geq 0 \ \ [R] \{\alpha\} \geq \{0\}
\end{align*}
\]

with the additional constraints

\[
either \ y_i = 0 \ or \ F_i = 0 \quad (18c)
\]

where \( \{z_1\} \) is a vector of \( n \) artificial variables and \( \{z_2\} \) is of \( m \) artificial variables. This problem is solved as a general linear programming problem with an exception to check on variables \( \{F\} \) or \( \{y\} \) when they enter as base variables in the calculation process.

When the matrix \( [S] \) is positive definite, the quadratic optimization problem, as defined in Eq.(13) and (14), has a unique solution (Cattle, 1966 and Kortanek and Jeroslow, 1967). Since \( [S] \) is always positive definite in structural problems, the contact problems has a unique solution from the optimiza-
tion process. The programming steps are shown in Table 1. The minimization of the artificial function leads to the basic feasible solution in a general linear programming problem (G.W. Dantzig, 1963). Since in this case, the problem has a unique solution, the solution arrived for the problem as stated in Eq.18 is the final one; no iterative cycle is necessary in this case.

FLEXIBILITY COEFFICIENTS FOR CONTACT POINTS

In the finite element analysis of a general 3-dimensional solid, the isoparametric elements are quite efficient and accurate in mathematical and geometrical idealisation (Zienkiewicz, 1971 and Clough, 1971). The isoparametric brick-type 10 node element with mid-side nodes is employed in this analysis; this element is capable of describing a quadratic interpolation function on its edges. The details of the element stiffness generation is detailed in the appendix. For any particular problem, the derivation of the flexibility coefficients for points in the possible region of contact is illustrated in Eq.(3) to (7).
### Variables

<table>
<thead>
<tr>
<th>( F_1 )</th>
<th>( F_2 )</th>
<th>( \cdots )</th>
<th>( \cdots )</th>
<th>( F_n )</th>
<th>( \alpha_1 )</th>
<th>( \alpha_2 )</th>
<th>( \alpha_3 )</th>
<th>( \cdots )</th>
<th>( \alpha_6 )</th>
<th>( y_1 )</th>
<th>( y_2 )</th>
<th>( y_3 )</th>
<th>( \cdots )</th>
<th>( y_n )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(-S_{11})</td>
<td>(-S_{12})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-S_{1n})</td>
<td>( R_{11} )</td>
<td>( R_{12} )</td>
<td>( R_{13} )</td>
<td>( \cdots )</td>
<td>( R_{16} )</td>
<td>1</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>2</td>
<td>(-S_{21})</td>
<td>(-S_{22})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-S_{2n})</td>
<td>( R_{21} )</td>
<td>( R_{22} )</td>
<td>( R_{23} )</td>
<td>( \cdots )</td>
<td>( R_{26} )</td>
<td>0</td>
<td>1</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>3</td>
<td>(-S_{31})</td>
<td>(-S_{32})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-S_{3n})</td>
<td>( R_{31} )</td>
<td>( R_{32} )</td>
<td>( R_{33} )</td>
<td>( \cdots )</td>
<td>( R_{36} )</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
</tr>
<tr>
<td>( n )</td>
<td>(-S_{n1})</td>
<td>(-S_{n2})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-S_{nn})</td>
<td>( R_{n1} )</td>
<td>( R_{n2} )</td>
<td>( R_{n3} )</td>
<td>( \cdots )</td>
<td>( R_{n6} )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>( n + 1 )</td>
<td>(-R_{11})</td>
<td>(-R_{12})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-R_{n1})</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>( n + 2 )</td>
<td>(-R_{21})</td>
<td>(-R_{22})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-R_{n2})</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
<td>( \vdots )</td>
</tr>
<tr>
<td>( n + m )</td>
<td>(-R_{m1})</td>
<td>(-R_{m2})</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-R_{nm})</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>( \cdots )</td>
</tr>
<tr>
<td>(-\varepsilon())</td>
<td>(-\varepsilon())</td>
<td>(-\varepsilon())</td>
<td>( \cdots )</td>
<td>( \cdots )</td>
<td>(-\varepsilon())</td>
<td>1</td>
<td>-1</td>
<td>-1</td>
<td>( \cdots )</td>
<td>-1</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Right Hand Side

| \( \sigma_1 + C_1 \) | \( \sigma_2 + C_3 \) |

### Last Row

- Last row is obtained by summing up the coefficients and changing sign. \( l \) - no. of points of applied forces.

### Programming Steps

1. Check for the lowest negative number in the last row and the corresponding column is \( r \).
2. Check for the lowest value of the ratio $R_i/b_i$ where $R_i$ is the value of the right side of the Eq. (i) and $b_i$ is the positive coefficient in $i$ th column. The corresponding $i$ th variable enters as a new variable.

3. If $F_i$ is entering the base, check whether $y_i$ is already in the base. If $y_i$ is not in the base, $F_i$ can enter; otherwise, if $F_i$ and $y_i$ are corresponding to the same row $i$, $F_i$ can enter; if they are not in the same row, $F_i$ can not enter; in the third case, steps 1 and 2 are repeated neglecting $i$ th column.

4. The above steps are repeated until the negative coefficients exist and a new variable can not enter the base.
IV NUMERICAL EXAMPLES.

1. Mathematical Programming Technique:

This optimization technique along with a finite element stiffness method forms a general formulation for elastic contact of solids. A computer program for this technique is developed as a part of requirement for the overall general formulation. This program is based on the Wolfe's modified simplex type linear programming technique which has been outlined in the general formulation.

A simple Hertzian problem of sphere to sphere contact is analysed by this technique as a test case.

Flexibility coefficients are obtained by employing the Hertz's approximation of solid into elastic half space for analysis. These coefficients are obtained with the use of Boussinesq's solution for a normal load on a semi-infinite medium; the corresponding deflection quantities at various points in the elastic half space are obtained by the following Boussinesq's solution.

\[ W_i = \frac{1-v^2}{\pi E} \frac{F_j}{d_{ij}} \]

Where \( d_{ij} \) is the distance from the point \( j \) where the force \( F_j \) is applied to the point \( i \) where the deflection \( W_i \) is measured; \( E \) is the young's modulus and \( v \) is the Poisson's ratio. The total deflection at the point \( i \) due to \( N \) discrete forces is

\[ W_i = \frac{1-v^2}{\pi E} \sum_{j=1}^{N} \frac{F_j}{d_{ij}} \]

Hence, the flexibility coefficient is

\[ a_{ij} = \frac{1-v^2}{\pi E} \frac{1}{d_{ij}} \]

The value of flexibility coefficient is infinity at the point of application of a force \( i=j \) as per Eq.(21). For our analysis, this value is arbitrarily taken to be twice that of the next closest discrete point due to that load. (Or the factor can be equal to the ratio between the flexibilities of two neighbouring points, separated as in the above problem but one point at the boundary of a uniformly loaded area and another in the unloaded area).
Once the overall flexibility matrix is developed for all probable contact points on the surface of the solids, the analysis is proceeded further by the mathematical programming technique.

Two test cases are solved by this technique; unequal spheres of 25 cm and 2.5 cm radii in contact due to a normal compressive load of 454 kg and secondly, equal radii spheres (25 cm) in contact for the same compressive load. In this analysis, a discrete force is assumed to represent the resultant pressure on a small domain of contact area. The results of the analysis are shown in the figs. 2 and 3. In the fig. 2a each discrete force represents the force distribution on an area of 0.0875 x 0.0875 sq.mm and in the fig. 2b for 0.075 x 0.075 sq.mm. The contact area is, hence, the shaded area as shown in the fig. 2. The stress variation along a diagonal of the circular area of contact is shown in fig. 3. From the results, it is seen that the area of contact evaluated by this technique is not well defined in comparison to the analytically predicted circular area. Also there are differences in the stress values between the two analyses. The small discrepancy in the stress values might be due to the incorrect evaluation of flexibility coefficients at those points where the discrete forces are applied and also where deflections are measured (i.e. when $d_{ij}=0$ in Eq.1). Comparison of the results in fig. 2 and 3, it is seen that convergent solution is possible to achieve by taking finer mesh (grid) divisions of the possible contact area.

The advantage of this method is that the computer programming is simple and also computer time requirement for an analysis is small unless many points are taken in the possible contact area. However, the contact area and the stress variations are not smooth since the analysis is carried out for discrete domains and forces. Another restriction is that the contact area must have at least a plane of symmetry about which deformation is symmetric for this method and for methods employing elastic half space results. This technique cannot be employed to the knee-joint analysis since the geometry does not have any plane of symmetry in the region of contact. As a test case when this technique applied to the knee geometry
Case 2a. Points are separated by a distance of 0.075 mm.

---

Case 2b. Points are separated by a distance of 0.075 mm.

- shows the boundary line of contact according to Herby analysis.
- discrete points in contact by the present mathematical programming analysis.

Fig. 2. Contact area for $P = 45.4$ kg. — Unequal radii of 85 and 35 cm.
Hertz's analysis for two spheres of 25 cm. in contact

disked points in contact

d points which are not in contact

<table>
<thead>
<tr>
<th>Herbo Analysis</th>
<th>Mathematical Program Technique</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid body approach</td>
<td>0.004013 m.m.</td>
</tr>
<tr>
<td>Maximum compressive stress</td>
<td>3964 kg/cm²</td>
</tr>
</tbody>
</table>

Fig. 2C. Contact area for P = 45.4 kg. - Equal radii of 25 cm.
Case 2a. Stress Variations

Case 2b. Stress Variations

--- pressure variations according to Henry analysis.
Θ --- case 2a.
Δ --- case 2b.

Results

\[
\begin{array}{ccc}
\alpha, \text{Rigid body approach} & \gamma, \text{error} & \text{maximum pressure at the centre} & \% \text{error} \\
\text{Timoshenko (Henry)} & 0.007075 \text{m.m.} & - & 12350 \text{kg/cm}^2 & - \\
\text{Case 2a.} & 0.00613 \text{m.m.} & 4.14 & 13310 \text{kg/cm}^2 & 7.83 \\
\text{Case 2b.} & 0.006125 \text{m.m.} & 3.53 & 13210 \text{kg/cm}^2 & 6.97 \\
\end{array}
\]

Fig. 3. Unequal Radii — Stress Variations.
assuming the plane of contact between the femur and tibia to be the tangent plane of the first contact point. The results are not correctly evaluated.

2. CONTACT ANALYSIS BY THE FINITE ELEMENT SYSTEM

In the Marc programming system, it is possible to combine contact elements with other type of elements in the pack to analyse contact behaviour of any two solids. The contact elements are introduced in the possible region of contact as shown in fig. 4.

![Figure 4](image)

The analysis is done by an incremental manner in this case.
The load is applied incrementally and at each increment of load, the stiffness matrix is calculated, assembled and solved for displacements and stresses.
The stress-strain diagram for a contact element is as shown in fig. 5.
In that figure, the stress-strain diagram has a break point when the compressive strain reaches a value of -1 and after that point, the element

![Figure 5](image)
Fig. 6 Discretization for the 'MARC'
System Data
does not take any strain practically; before the break point, the element has large strain even for a small value of stress. This property is employed in the contact analysis suitably as follows. Until when a pair of points connected by a contact element comes in contact, the element deforms greatly even for a small load as shown in the fig. 5, with a small value of Young's modulus for strains more than -1.0. When the two points come in contact, the compressive strain in the contact element reaches a value of -1.0 and after that the contact element does not take any more strain because of its very high Young's modulus and only the solids have to deform hereafter. In this analysis, a very small increment of load is applied each time and the increment should not be very high so that the contact elements do not deform more than -1.0 compressive strain due to the low modulus of elasticity assumed for the contact elements during that increment. In the MARC system, it is also possible to have iterative cycles for each increment thus finding average stiffness values for the elements to calculate the deflection; during the second cycle for the same increment the stiffness values of contact elements can change due to its stress-strain behaviour. For high incremental loading, the iteration for average stiffness values yields a better solution. However, it is desirable to find a suitable way involving iteration and incremental loading through the RESTART option which is available in the system.

The problem solved by this MARC system analysis is a test problem which is a simple Hertzian equal radiisphere-sphere contact. Quadrilateral axisymmetric elements are employed to discretize the spheres and contact elements are introduced in the possible region of contact as shown in fig. 6. Due to symmetry, only a symmetrical half of the contact problem as shown in fig. 6 is analysed each time. A list of the MARC data is included at the end of this report.

iii) General formulation.

This formulation is employed to the same test problem of equal radii sphere to sphere, contact pressed together by a compressive force of 726.4 kg. The finite element discretization is as shown in fig. 7. Finer mesh points are required in the region of contact since the contact area is small and also the stress variation within the area is appreciable. The
finer mesh discretization is achieved by the gradually decreasing size elements as in fig. 7.

In this problem due to symmetry conditions, there is only a vertical rigid body displacement possible and it is to be removed or restricted by prescribing or arbitrary support condition in the vertical direction at a point on the symmetrical axis.

The 3-dimensional 20 node isoparametric elements are employed in this problem to generate stiffness matrix of the bodies in contact. The stiffness coefficients at the mesh points of the possible region of contact are reduced from the overall structural stiffness matrix as shown in Fig. (3) and (4). After assembling for the whole structure, the stiffness matrix is partitioned as in Fig. 3. The possible contact points are indexed and the stiffness matrix is reduced. Even though nodal numbering of the points of the contact area can be done anyway, it may be numbered in the end for easier reduction of the structural stiffness matrix.

\[
\begin{bmatrix}
K_{NN} & K_{Ne} \\
K_{eN} & K_{ee}
\end{bmatrix}
\begin{bmatrix}
U_N \\
U_e
\end{bmatrix} =
\begin{bmatrix}
F_N \\
F_e
\end{bmatrix}
\]

\[\text{eqn. (3a)}\]

\[
\begin{bmatrix}
\bar{K}_{ee}
\end{bmatrix}
\begin{bmatrix}
U_e
\end{bmatrix} = 
\begin{bmatrix}
\bar{F}_e
\end{bmatrix}
\]

\[\text{eqn. (3b)}\]

where

\[
\begin{bmatrix}
\bar{K}_{ee}
\end{bmatrix} =
\begin{bmatrix}
K_{ee}
\end{bmatrix} -
\begin{bmatrix}
K_{eN}
\end{bmatrix}
\begin{bmatrix}
K_{NN}
\end{bmatrix}^{-1}
\begin{bmatrix}
K_{Ne}
\end{bmatrix}
\]

\[\text{eqn. (4a)}\]

\[
\begin{bmatrix}
\bar{F}_e
\end{bmatrix} =
\begin{bmatrix}
F_e
\end{bmatrix} -
\begin{bmatrix}
K_{eN}
\end{bmatrix}
\begin{bmatrix}
K_{NN}
\end{bmatrix}^{-1}
\begin{bmatrix}
F_N
\end{bmatrix}
\]

\[\text{eqn. (4b)}\]

where in \(\bar{K}_{ee}\) is the reduced stiffness matrix for possible contact points; \(\{\bar{F}_e\}\) is the force vector acting in the contact area and which is to be evaluated by the optimization technique.

Due to symmetry, only a quadrant of the sphere is taken for analysis the lower sphere as shown in fig. 8 is assumed to be fixed at the support points and so the finite element stiffness analysis is performed in the
usual way. In the case of the upper sphere where no support condition
is prescribed, the stiffness matrix is singular and the inversion is
possible only when the rigid body motion in the vertical direction is
restrained; for this case, any point on the symmetrical axis can be fixed
to restrain that rigid body motion and the analysis can be performed as
usual. The analysis is performed in two ways: 1) by fixing up the lowest
point on the vertical axis in the contact region itself 2) by fixing
up the central point of the vertical axis as shown in fig. 8. However
final results in both cases should be same; the analysis is checked for
these two cases to know whether introduction of a support condition at
the possibly smaller contact area will cause numerical difficulties
and hence, incorrect results.

After evaluating the reduced stiffness coefficients, the mathematical
programming technique is employed to arrive at the unique solution a-
available for this contact problem. The evaluated discrete forces in
the contact area in both cases are exactly same except for the rigid
body motions of the assumed supports. From the known contact forces
\( \{ F_e \} \), the displacements in the sphere are calculated by the back sub-
stitution in the stiffness matrix as in Eq. (3) and then the stress
values at the required points are evaluated. The stress values obtained
from this formulation are shown in fig. 9. From the results, it is seen
that the maximum stress value arrived at the end of this analysis by
by this formulation is less than the analytically predicted result.
Tensile stresses as shown in the outer boundary of the quadrant cir-
ple in fig. 9 are not correct values since the jacobian at those points
are singular (since the tangent is common for two sides of those ele-
ments).

When the load is less, the contact area is very small and it is diffi-
cult to get reasonable values by this formulation because of numerical
difficulties. In the first instance, this formulation is employed to
the same problem with a lesser load \( P = 45.4 \text{ kg.} \) and the results are
not correct. This is mainly due to the difficulty of analysing a very
small region in the analysis. For instance, for the load \( P = 45.4 \text{ kg.} \),
the ratio of the linear dimension of the contact area to the radius of the
sphere as per the analytical prediction is.
Trying to analyse such a small region by discretizing into elements and analysing by finite element method do not give correct values in this case. Hence the load is increased in the second time. With $P = 726.4$ kg. the analysis is carried out and the corresponding results are shown in fig. 9. Even in this case, the ratio of radius of contact circle to the radius of the sphere is $1/140$. The analysis of such a small region may be the reason for the difference in stress values obtained by this method to those by analytical Hertzian theory.

However this formulation can be very well applied for any shape of bodies in contact and without the restriction that the contact area is small compared to the size of the contacting solids. In fact, it can be employed without difficulty for problems where contact area is appreciable.
- 42 elements
- 338 nodes
- 24 points in the contact

Fig. (71). Discretization of a quarter sphere

Figuur 7b. Discretization about a section -AB section
Spetry conditions
Symmetry conditions

$E = 2.0 \times 10^6 \text{ kg/cm}^2$
$v = 0.3$

Case 1. Point 'A' is supported
Case 2. Point 'B' is supported

Fig. 8. Sphere-sphere contact
Fig. 9. Equal radii spheres in contact
- contact forces and stresses (compressive)-
load = 726.4 kgm.

Rigidbody approach $\alpha$
case (i) artificial support at the centre of contact area
$\alpha = 0.04895 \text{ mm}$
case (ii) artificial support at the centre of the sphere
$\alpha = 0.061305 \text{ mm}$

Heriz's analysis $\alpha = 0.02548 \text{ mm}$
Radius of contact circle $= a = 1.7845 \text{ m}$
Compressive stress at the $\sigma_{zz} = 9990 \text{ kg/cm}$
centre of contact area
V. REFERENCES


VI APPENDIX A

SEPERATION OF RIGID BODY MOTIONS FOR ELASTIC ANALYSES;

In the case of finite element elastic analysis, the equilibrium equation is written as

$$\{F\} = [K] \{u\} \quad (A-1)$$

where \(\{u\}\) is the total displacement vector having both elastic and rigid body displacements of the concerned body. When support conditions are prescribed, it is easier to eliminate the rigid body components. However, in the case of contact analysis, when a body is supported by force equilibrium and but no kinematic boundary conditions, as shown in fig. A-1, for a finite element analysis \([K]\) is a singular matrix. In such a case, elastic analysis can be carried out with arbitrary support conditions at certain points in the body. In a contact analysis, elastic displacements also being important for final results, the assumed supports are given rigid body motions to keep the problem to be the same. Another way of treating this case is illustrated by Veldpaus (1976)

The displacement vector \(\{u\}\) is separated into elastic displacement and rigid body contributions as follows.

$$\{u\} = [S] \times \{U_e\} + [R] \{\alpha\} \quad (A-2)$$

where \(N_r\) is the number of rigid body motions present in the analysis;
\([\{S\}]\) and \([\{U_e\}]\) are matrices representing elastic displacement contributions and \([\{R\}]\) and \([\{a\}]\) are for rigid body motions.

\[
\{U\} = [S \, R] \begin{bmatrix} U_e \\ a \end{bmatrix}
\]  \hspace{1cm} (A-3)

Substituting Eq. (A-3) in Eq. (A-1) and premultiplying with \([S^T\]) matrix, we get

\[
\begin{bmatrix} S^T \\ R^T \end{bmatrix} \begin{bmatrix} \{K\} & \{S\} & \{R\} \end{bmatrix} \begin{bmatrix} U_e \\ a \end{bmatrix} = \begin{bmatrix} S^T \\ R^T \end{bmatrix} \{F\}
\]  \hspace{1cm} (A-4)

Rewriting

\[
\begin{bmatrix} S^T K & S^T K K & \{U_e\} \\ R^T K & S^T K R & \{a\} \end{bmatrix} = \begin{bmatrix} S^T F \\ R^T F \end{bmatrix}
\]  \hspace{1cm} (A-5)

Since rigid body motions do not generate external forces,

\[
\{K\} \{R\} \{a\} = 0
\]

so:

\[
\{K\} \{R\} = 0
\]  \hspace{1cm} (A-6)

Substituting Eq. (A-6) in Eq. (A-5), we get:

\[
\begin{bmatrix} S^T K & 0 \\ 0 & 0 \end{bmatrix} \begin{bmatrix} U_e \\ a \end{bmatrix} = \begin{bmatrix} S^T F \\ R^T F \end{bmatrix}
\]  \hspace{1cm} (A-7)

or:

\[
[S]^T \{K\} [S] \{U_e\} = [S]^T \{F\}
\]  \hspace{1cm} (A-8a)

\[
[R]^T \{F\} = 0
\]  \hspace{1cm} (A-8b)

From the Eq. (A-8a) elastic displacement vector \([U_e]\) is

\[
\{U_e\} = ([S]^T [K] [S])^{-1} [S]^T \{F\}
\]  \hspace{1cm} (A-9)

or:

\[
\{U_e\} = [K_e]^{-1} [S]^T \{F\}
\]

Inversion of \([S]^T [K] [S]\) is possible since the matrix is not singular anymore. Substituting Eq. (A-9) in Eq. (A-2), we get the total displacements
In a contact situation, Eq (A-10) is partitioned to get the total displacements \( \{ \mathbf{U}_e \} \) in the possible region of contact.

\[
\{ \mathbf{U}_e \} = \left[ S_e \right] \left[ K_e \right]^{-1} \{ S_e \}^T \{ F_e \} + \left[ R_e \right] \{ \alpha \} \quad (A-10)
\]

So we get:

\[
\begin{bmatrix}
\mathbf{U}_N \\
\mathbf{U}_e
\end{bmatrix} = \left[ S_e \right] \left[ K_e \right]^{-1} \left[ S_e \right]^T \{ F_e \} + \left[ S_e \right] \left[ K_e \right]^{-1} \left[ S_e \right]^T \{ F_e \} + \left[ R_e \right] \{ \alpha \} \quad (A-11)
\]

From Eq. (A-12b),

\[
\begin{bmatrix}
\mathbf{R}_e^T \\
\mathbf{R}_e^T
\end{bmatrix} \{ F \} = 0 \quad \text{or} \quad \mathbf{R}_e^T \{ F \} = - \left[ \mathbf{R}_e \right]^T \{ F \} \quad (A-13)
\]

which states that the contact forces are in equilibrium with the forces applied elsewhere in the body but in opposite direction.

In the Eq. (A-12), the first term of the right hand side is a known quantity and the second and third terms involve \( \{ F_e \} \) and \( \{ \alpha \} \) which are to be evaluated in the contact analysis at a later stage.

This way of separation of rigid body motions is a general approach. However, the matrix \([ S \, R ]\) as shown in Eq. (A-3), is to be evaluated as a first step to employ this approach; this evaluation is outlined as folows.

This illustrated by a 2-dimensional example as shown in fig. (A-2). A suitable point in the body is taken as origin and about the origin, the rigid body motions \( u, v \) and \( \phi \) are applied on the body.

After discretizing this body into finite elements, the matrix \([ R ]\) in the
Eq. (A-3) is generated as follows:

\[
[R] = \begin{bmatrix}
1 & 0 & y_1 \\
0 & 1 & -x_1 \\
1 & 0 & y_2 \\
0 & 1 & -x_2 \\
\vdots & \vdots & \vdots \\
0 & 1 & -x_n
\end{bmatrix}
\]

\[
\alpha = \begin{bmatrix} u \\ v \\ \phi \end{bmatrix}
\]

(A-14)

The matrix [R] is normalised and [S] is generated from the matrix [R] with the following conditions:

let

\[
[R] = [\bar{r}_1 \bar{r}_2 \bar{r}_3]
\]

\[
[S] = [\bar{s}_1 \bar{s}_2 \bar{s}_3 \ldots \bar{s}_{2n-3}]
\]

(A-15)

where \(\bar{r}_i\) and \(\bar{s}_i\) are column vectors

\[
-s_j^{T} \bar{r}_i = 0 \quad j = 1, 2, \ldots 2n-3 \text{ and } i = 1, 2, 3,
\]

(A-16)

\[
-s_i^{T} \bar{s}_j = \delta_{ij} \quad i, j = 1, 2, 3, \ldots 2n-3
\]

\[
-r_i^{T} \bar{r}_j = \delta_{ij} \quad i, j = 1, 2, 3
\]

where \(\delta_{ij}\) is kronecker delta.
With these conditions, Gram-Schmidt orthogonalization technique is employed to develop the matrix \([S]\). In dynamics problems, another condition is introduced to generate \([S]\)(which is uniquely obtained only with this condition).

\[
[S]^T [M] [R] = 0
\]  \hspace{1cm} (A-17)

where \([M]\) is the mass matrix.
VII. Appendix B

ISOPARAMETRIC THREE DIMENSIONAL 20 NODE HEXAHEDRON ELEMENT

INTRODUCTION

Three dimensional finite element analysis is employed in engineering mechanics problems somewhat easily by the introduction of the isoparametric element family by Zienkiewicz and his colleagues (Irons, 1966, Ergatoudis, 1966, Irons and Zienkiewicz, 1968). The advantage of the isoparametric hexahedron element is that any complex geometry can be easily discretized and well approximated; the element stiffness matrix is evaluated easily and automatically by assuming for the displacement function, the same interpolation function which transforms the local coordinates of the element to its global cartesian coordinates. A comprehensive survey of the 3 dimensional elements is presented by Clough (1969).

In this report, the stiffness derivation for the 20 node isoparametric element and shape function modifications required as the nodes of one edge coalesce to a single node are detailed. A computer programming list for the element stiffness generation is also included at the end.

Element Stiffness Calculation

Local coordinates of the hexahedron element are defined by its natural coordinates which are having magnitudes of ±1 for any pair of opposing faces of a cube. These local coordinates are transformed to the global cartesian coordinate system by quadratic interpolation function that are defined along the element edges. This transforms a cubic element (in local coordinate system) to a distorted hexahedron with its element edges quadratically curved as shown in fig. (8-1)
a) Local coordinates, \((\xi, \eta, \zeta)\)  

b) Cartesian global coordinates, \((x, y, z)\)

**fig. a.1 ELEMENT DETAILS**

The relation between the local and cartesian coordinates are uniquely defined by the following transformation.

\[
\begin{bmatrix}
 x \\
 y \\
 z
\end{bmatrix} = \begin{bmatrix}
 x_1, x_2, x_3, \ldots, x_{20} \\
 y_1, y_2, y_3, \ldots, y_{20} \\
 z_1, z_2, z_3, \ldots, z_{20}
\end{bmatrix} \begin{bmatrix}
P_1 \\
P_2 \\
P_3 \\
\vdots \\
P_{20}
\end{bmatrix}
\]

(B-1)

where \(P_i\) \((i=1,2,\ldots,20)\) are interpolation function in local coordinates.

\[
P_i = (1+\xi_i)(1+\eta_i)(1+\zeta_i)(\xi_i\eta_i\zeta_i-2)/8
\]

\[(B-2)\]

for \(i=1,2,\ldots,8\) corner nodes.

\[
P_i = (1+\xi_i)(1+\eta_i)(1+\zeta_i)(1-\xi_i^2 - \eta_i^2 - \zeta_i^2 + \xi_i^2 \eta_i^2 + \xi_i^2 \zeta_i^2 + \eta_i^2 \zeta_i^2 - \xi_i^2 \eta_i^2 \zeta_i^2)/6
\]

\[(B-3)\]

for \(i=9,10,\ldots,20\) mid side nodes.

\(x_i, y_i, z_i\) \((i=1,2,\ldots,20)\) are the nodal coordinate values of the element in the global coordinate system. \(\xi, \eta, \zeta\) are local coordinates of any point on the element and \(\xi_i, \eta_i, \zeta_i\) \((i=1,2,\ldots,20)\) are nodal values. This ensures continuity of the displacement function at the common points of two adjacent elements. Rewriting the Eq. (B-1).
where \( P = \{ P_1, P_2, P_3, \ldots, P_{20} \} \)

\[
{x'}^T = \{ x_1, x_2, x_3, \ldots, x_{20} \}
\]

The displacement function is assumed to be the same interpolation function generated in consideration of the geometry as in Eq. (B-4). Hence, we get

\[
\begin{bmatrix}
\mathbf{u} \\
\mathbf{v} \\
\mathbf{w}
\end{bmatrix} =
\begin{bmatrix}
P & 0 & 0 \\
0 & P & 0 \\
0 & 0 & P
\end{bmatrix}
\begin{bmatrix}
\mathbf{u} \\
\mathbf{v} \\
\mathbf{w}
\end{bmatrix}
\]

(B-6a)

where \( \mathbf{u}, \mathbf{v}, \mathbf{w} \) are the nodal displacements which are unknowns and are to be evaluated.

\[
\begin{aligned}
\mathbf{u}^T &= \{ u_1, u_2, \ldots, u_{20} \} \\
\mathbf{v}^T &= \{ v_1, v_2, \ldots, v_{20} \} \\
\mathbf{w}^T &= \{ w_1, w_2, \ldots, w_{20} \}
\end{aligned}
\]

(B-6b)

The general strain quantities in 3-dimensional analysis are obtained by employing the Eq. (B-6a) as follows.

\[
\{ \mathbf{E} \} = [B] \{ d \}
\]

(B-7b)
By the chain rule of differentiation, we get

\[
\begin{bmatrix}
\frac{\partial P}{\partial \xi} \\
\frac{\partial P}{\partial \eta} \\
\frac{\partial P}{\partial \zeta}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\
\frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta}
\end{bmatrix} \begin{bmatrix}
\frac{\partial P}{\partial x} \\
\frac{\partial P}{\partial y} \\
\frac{\partial P}{\partial z}
\end{bmatrix} = [J] \begin{bmatrix}
\frac{\partial P}{\partial x} \\
\frac{\partial P}{\partial y} \\
\frac{\partial P}{\partial z}
\end{bmatrix}
\] (B-8)

Hence,

\[
\begin{bmatrix}
\frac{\partial P}{\partial x} \\
\frac{\partial P}{\partial y} \\
\frac{\partial P}{\partial z}
\end{bmatrix} = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\
\frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta}
\end{bmatrix}^{-1} \begin{bmatrix}
\frac{\partial P}{\partial x} \\
\frac{\partial P}{\partial y} \\
\frac{\partial P}{\partial z}
\end{bmatrix}
\] (B-9)

where \([J]\) is the jacobian matrix of coordinate transformation and can be evaluated easily when the same interpolation function is used to describe geometry and displacement. Using the Eq. (B-1) and rearranging the matrix elements, the Jacobian matrix, \([J]\) is evaluated as follows.

\[
[J] = \begin{bmatrix}
\frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\
\frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\
\frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta}
\end{bmatrix}
\] (B-10)

The strain matrix is given in terms of local coordinates as shown in Eqs. (B-7a) and (B-9); hence, the integration for the element stiffness calculation is also done in the local coordinates. Once the strain matrix is established, the stiffness evaluation is proceeded as it is done in the general finite element analysis.

\[
[K] = \int \int [B]^T[D][B] \, dv = \int \int \left[\begin{array}{l}
1 \\
1 \\
1
\end{array}\right]^T \left[\begin{array}{l}
dx \\
dy \\
dz
\end{array}\right] [D] \left[\begin{array}{l}
dx \\
dy \\
dz
\end{array}\right] \, d\xi d\eta d\zeta
\] (B-11)

where the differential volume \(dv\) is
\[
dv = dx \, dy \, dz = d\xi \, d\eta \, d\zeta /J/
\]

where \(J\) is the determinant of the Jacobian matrix. The elasticity matrix \([D]\) is given as follows.

\[
[D] = \frac{E}{(1+v)(1-2v)} \begin{bmatrix}
1-v & v & 0 & 0 & 0 \\
v & 1-v & v & 0 & 0 \\
v & v & 1-v & 0 & 0 \\
0 & 0 & 0 & \frac{1-2v}{2} & 0 \\
0 & 0 & 0 & 0 & \frac{1-2v}{2}
\end{bmatrix}
\]

where \(E\) is Young's modulus and \(v\) is Poisson's ratio.

The Eq. (b-11) is numerically integrated by the Gaussian quadrature method to obtain the element stiffness method.

**Corrections in shape functions to account for Degenerated Element.**

It has been shown that extensive modification in the shape function is required where an elemental edge coalesces to a single node in the case of isoparametric element employing quadratic interpolation displacement function along its edges (Newton, 1974 and Irons, 1974). The step by step study of degeneration would not be possible due to the coalescing nodes. However, it has been shown that the elements employing linear interpolation function along their edges would not require any correction and the respective stiffness contributions of the coalescing nodes could be summed up. In the case of isoparametric elements employing quadratic interpolation functions along their edges, summing up stiffness contributions at the coalescing nodes alone does not account fully for the degenerated final shape. This is illustrated by a simple two dimensional degenerating quadrilateral isoparametric element.

Fig. (B-2) shows the degeneration of the quadrilateral element when one of its edge coalesces to a point and modifies as a triangular element.
The assumed shape functions along the edges of this element are

\[ P_i = (1 + \xi \eta_i)(1 + \eta_i)(\xi \eta_i + \eta_i^2 - 1)/4 \quad i=1,2,3,4 \text{ corner nodes.} \]

\[ P_i = (1 + \xi \eta_i)(1 + \eta_i)(1 - \xi^2 \eta_i^2 - \eta_i^2)/2.0 \quad i=5,6,7,8 \text{ midside nodes.} \]

The triangular element shown in fig. (B-2b) is a well known element (Zienkiewicz, 1971) representing quadratic displacement variations along its sides and the displacement function in area coordinates \((L_1,L_2,L_3)\) which are evaluated as follows.

\[
\begin{align*}
L_1 &= \frac{\text{Area of } \Delta P_{23}}{\text{Area of } \Delta 123} \\
L_2 &= \frac{\text{Area of } \Delta P_{13}}{\text{Area of } \Delta 123} \\
L_3 &= \frac{\text{Area of } \Delta P_{12}}{\text{Area of } \Delta 123}
\end{align*}
\]

Fig. B-3. Area Coordinates.

These area coordinates are related to the coordinates \(\xi, \eta\) shown in fig. B-2a, as follows

\[
\begin{align*}
L_1 &= (1-\xi)(1-\eta)/4 \\
L_2 &= (1+\xi)(1-\eta)/4 \\
L_3 &= (1+\eta)/2
\end{align*}
\]

The assumed shape functions for this triangular element are

\[
P_i = (2L_i - 1)L_i \quad \text{for corner nodes} \\
P_i = 4L_i L_j \quad \text{for midside node at the side } ij
\]
The coalesced quadrilateral element shown in fig. B-2b, must have the same interpolation displacement function as that of the triangular element; the interchangability is not automatically achieved on degeneration and modifications in the shape functions are required in the following nodes as given below:

\[ P_1^c = P_1 + \Delta P \]
\[ P_2^c = P_2 + \Delta P \]
\[ P_5^c = P_5 - 2\Delta P \]

where \( \Delta P = (1-\xi^2)(1-\eta^2)/8 \)

At the coalescing node
\[ P_3^c = P_3 + P_4 + P_7 \]

and at other two nodes
\[ P_8^c = P_8 \] and \[ P_6^c = P_6 \]

In the similar manner, the hexahedron element employing quadratic interpolation function along its edges also needs modifications in the displacement function when the element edges are coalescing to a node. The nodes requiring modifications in the interpolation functions are indicated in fig. (B-4b) and (B-4c) when the element is degenerating as shown in those figures.

Fig. B-4 a) hexahedron b) with one coalesced edge c) with two coalesced edges.
The nodes shown in fig. B-4b require correction in the shape function as follows

\[ P_7^c = P_7 + P_{19} + P_8 \quad (B-18a) \]

\[ P_3^c = P_3 + \Delta P_1; P_4^c = P_4 + \Delta P_1; P_{11}^c = P_{11} - 2\Delta P_1 \quad (B-18b) \]

\[ \Delta P_1 = (1 - \eta^2)(1 - \zeta^2)(1 + \xi)/16 \quad (B-18c) \]

\[ P_5^c = P_5 + \Delta P_2; P_6^c = P_6 + \Delta P_2; P_{17}^c = P_{17} - 2\Delta P_2 \quad (B-18d) \]

\[ \Delta P_2 = (1 - \xi^2)(1 - \eta^2)(1 + \xi)/16 \quad (B-18e) \]

The degenerate form shown in fig B-4c (Prism element requires modifications at the following nodes

\[ P_3^c = P_3 + P_{11} + P_4; P_7^c = P_7 + P_{19} + P_8; P_{15}^c = P_{15} + P_{16} \quad (B-19a) \]

\[ P_1^c = P_1 + \Delta P_1; P_2^c = P_2 + \Delta P_1; P_9^c = P_9 - 2\Delta P_1 \quad (B-19b) \]

\[ \Delta P_1 = (1 - \xi^2)(1 - \eta^2)(1 - \zeta)/16 \quad (B-19c) \]

Corrections at the nodes 5, 6 and 7 are same as in Eq. (18d) and (18e).
NUMERICAL EXAMPLES ON THE APPLICATIONS OF THIS ELEMENT

This 3 dimensional element is tested for the following problems.

i) A simply supported beam with a uniformly distributed loading as shown in fig. B-5.

ii) Curved beam with internal pressure (Fig. B-6).

iii) Circular plate with central point load (Fig. B-8)

This element is also employed to an idealised total hipprosthesis having three components of varying material properties, femur bone, bone cement and stainless steel implant, as shown in fig. B-9.

A simply supported beam with a uniformly distributed loading is analysed to test the correctness of the element stiffness matrix. The discretization, the loading and the results are shown in the fig. B-5. The results agree very well with the beam theory. The stress $\sigma_{zz}$ is very sensitive to small changes in the deflection values. When ten elements are taken along the lengthwise direction the values are converging closely to the analytical value.

Case ii)

A simple problem of thick circular cylindrical shell with internal pressure is analysed to study the element size effects in the convergence of finite element solution. Analyses are made with a single element along a quadrant of the cylindrical shell and also with widely
varying element sizes as shown in fig. B-6.

(a) Equal thickness elements

(b) Unequal thickness elements.

Unit length of cylinder

Fig. B-6 Circular cylindrical shell with internal pressure.

(c) Unequal thickness elements -2 elements in the quadrant.

From the results as shown in fig. B-7, it seems that there are not much varying results due to discretization error even when the quadrant is approximated with a single element or when unequal elements are connected as in the fig. (B-6b) and (B-6c). However, the circumferential stresses are closer to the analytical results than the radial stresses as shown in Fig (B-7).
Thick cylinder - internal pressure
- analytical values
Θ 2 elements of equal thickness
△ 2 elements of unequal and contrasting thicknesses
☐ 4 elements - 2 elements along the quadrant of the cylinder - unequal thickness elements.

Fig. B-7 Internal Pressure on a thick cylinder: study of element size effects
Case iii)

A simply supported circular plate with a concentrated point load at the center of radius is analysed with and without prism elements generated by the program to investigate the degeneration behaviour of this isoparametric element to a prism element. The discretizations and results are shown in fig. (B-8). Results due to the finite element analyses are somewhat lower than the analytically predicted values; further mesh refinement may be necessary in this problem. However, these finite element analyses are shown to be closely agreeing.

![Diagram of a circular plate with point load](image)

\[ P = 100 \text{ kg} \]
\[ E = 2 \times 10^6 \text{ kg/cm}^2 \]
\[ v = 0.3 \]

a) Analysis with degenerated elements.

b) Analysis without degenerated elements.

**Deflection at A.**

I) Plate theory \(-0.02121 \text{ cm}\)

II) F.E. analysis (fig. B-8a) \(-0.0169 \text{ cm} \quad -0.0136 \text{ cm}\)

III) F.E. analysis (fig. B-8b) \(-0.0175 \text{ cm} \quad -0.0144 \text{ cm}\)

IV) F.E. analysis (fig. B-8c) \(-0.0166 \text{ cm} \quad -0.0139 \text{ cm}\)

**Fig. B-8 STUDY OF DEGENERATION BEHAVIOUR**

C) Analysis with vanishing Jacobian at point C.

In fig. (B-8C), the first element has two sides on a quadrant of a circle and so the corner nodal point C has a vanishing Jacobian (eq. B-8). However, when the stiffness is calculated by numerical integration, singularity at the boundary point does not influence deflection values as shown in the fig. B-8. However, when stress values at the point C is calculated numerically, (taking Jacobian at point C), the results will be incorrect, for instance, on compari-
son of the stress values of the analyses shown in fig (B-8b) and (B-8c), there exist considerable differences in their values at the point C.

<table>
<thead>
<tr>
<th>Stresses</th>
<th>AT 'A'</th>
<th>AT 'B'</th>
<th>AT 'C'</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>fig. 8b</td>
<td>fig. 8c</td>
<td>fig. 8b</td>
</tr>
<tr>
<td>$\sigma_{xx}$</td>
<td>-20297</td>
<td>-19160</td>
<td>-7491</td>
</tr>
<tr>
<td>$\sigma_{yy}$</td>
<td>-20297</td>
<td>-19160</td>
<td>-4894</td>
</tr>
<tr>
<td>$\sigma_{zz}$</td>
<td>-10330</td>
<td>-10179</td>
<td>391</td>
</tr>
</tbody>
</table>

Fig. (B-8d) Stresses on top surface of the plate - in kg/cm$^2$

Eventhough introduction of element such as '1' in fig (B-8c) does not alter the deflection behaviour, it may be avoided in view of incorrect stress values at certain points.

This isoparametric element has been employed to analyse total hip prosthesis having three components of different materials, femur bone, bone cement and stainless steel implant. The discretization of the implant is as shown in fig B-9. This analysis gives very good results and the are shown in comparison with axisymmetric element analysis of this same problem elsewhere by Huiskes.
Fig B-9 (a) idealised total hip implant (b) finite element discretization
Fig. B-9 (c) finite element discretization on the section AB
(half symmetry is assumed in the analysis).
A list of the computer program to analyse a general contact problem (without friction in the contact interface) is included in this report. The program is run in the computer in four steps:

I. Calculation of element stiffness matrices and storing in a disc or tape unit. The unit no. 2 is selected in this case; since the stiffness generation takes a long computer time, it is advisable to save them for further analysis.

II. Assembling and reducing the stiffness matrix to the required points in the possible contact region. The nodal numbering is done such a way that those points in the contact region are numbered at the end.

III. Modification of the reduced stiffness matrix to satisfy the problem requirements, for example, the stiffness matrix is further reduced by matrix condensation process, along a particular direction when rigid body movement is allowed in that direction only; in case of arbitrary contact shape like knee joint, the stiffness matrix is transformed to the directions of local normals of pairs of contacting points.

After modification of the stiffness matrix, it is augmented with other elements as shown in table 1 (page 18) and optimization program is run. Discrete contact forces are evaluated.

IV. Knowing the contact forces, a normal finite element program is run for displacements and stresses in the solids.

STEP. 1

Some points, which are to be taken care of, are described here

- **NELCOM** = 0 when the program is run for the first time; for example, if already 5 records are written, **NELCOM** = 5 and so following run, the next record is written as 6th one.

- **NCONT** = 0, then program calculates only element stiffness matrices and stores them and ends with those calculations. If **NCONT** = 200 (for example), no element stiffness matrix is calculated, but the program assembles the overall stiffness matrix and condenses the matrix to
the nodes from 200 to the maximum nodal number (say, 230); that is, the stiffness matrix corresponding to these 31(200-230) nodes are obtained by matrix condensation.

Hence, for the first step, NCONT = 0.

- **NTYPE:** if elements can not be grouped into definite sets, NTYPE = NELEM.

- **PLD(24)** the pressure load acts on the shaded side of the element. Hence,

  \[
  \text{NLEM (J,1)} = 1 \\
  \text{NLEM (J,2)} = -1 
  \]

     \[\text{Ith element}\]

Always, nodal numbering is given as shown in the figure according to the value of \text{NLEM(J,2)}, the nodal numbers of the concerned side is calculated. For example, the order in this case is: 1, 2, 5, 6, (first corner nodes), 9, 13, 14, 17. In the same order, the pressure intensities at the nodes are given as data in \text{PLD} array. \text{PLD(1)} to \text{PLD(8)} for \(\xi\) direction \text{PLD(9)} to \text{PLD(16)} for \(\eta\)-direction and \text{PLD(17)} to \text{PLD(24)} for \(\zeta\)-direction. In this example, \text{PLD} array has the following values:

\[
\begin{matrix}
+1.0 & +1.0 & +1.0 & +1.0 & +1.0 & +1.0 & +1.0 & +1.0 & \rightarrow \xi \\
0.0 & 0.0 & 0.0 & 0.0 & 0.0 & 0.0 & 0.0 & \rightarrow \eta \\
0.0 & 0.0 & 0.0 & 0.0 & 0.0 & 0.0 & 0.0 & \rightarrow \zeta
\end{matrix}
\]

corresponding to the following nodes

\[1 \quad 2 \quad 5 \quad 6 \quad 9 \quad 13 \quad 14 \quad 17\]

The step 1 is completed with stiffness generation and storing the matrices in an assigned unit (2).

**STEP. 2**

When \text{NCONT} is other than zero, the step 2 is carried out as explained earlier. In this step, matrix condensation is applied to the overall geometric matrix (as shown in Eq. 3 and 4) to arrive at the stiffness equations for the contact case.

**STEP. 3**

This step is consisting of reducing the stiffness matrix further and
assembling and augmenting the reduced matrix as shown in table 1. This portion is mainly a problem dependent and so left to the user to form a main program according to the requirement. After obtaining the matrix as in table 1, the subroutine PROG is called to perform the optimization process and evaluating the contact forces at the nodal points.

STEP. 4
A general finite element analysis, along with these contact forces (additionally) to evaluate displacements and stresses is performed.
COMPUTER PROGRAM

The following list gives the program details to generate stiffness matrix and after evaluation of displacements with the help of FEM, to evaluate the stresses at nodes of each element. The subroutines (procedures) which are to be known for immediate usage are detailed in the following. Other subroutines are called within the following routines. The manner in which the nodes are numbered is shown in fig. 1.

I SUBROUTINE STIFF 3 (XE,E,PA,NGAUS,TI)

DIMENSION XE(20,3),TI(60,60)

XE - element nodal coordinates (in the same order of nodal numbering as in fig. 1): $x_1, y_1, z_1; x_2, y_2, z_2,..., x_{20}, y_{20}, z_{20}$

E - Young's modulus; PA - Poisson's ratio.

NGAUS - number of Gaussian integration points for one dimensional integration - can be 1 or 2 or 3 or 4. IF NGAUS = 2, in this case, the total integration points in the element is $2^3 = 8$.

TI - final element stiffness matrix.

[TI] [u] = [F]

The corresponding displacement vector [u] is

$\{u\}^T = \{u_1, v_1, w_1, u_2, v_2, w_2,..., u_{20}, v_{20}, w_{20}\}$ and the force vector is

$\{F\}^T = \{f_1, x, f_1, y, f_1, z, f_2, x, f_2, y, f_2, z,..., f_{20}, x, f_{20}, y, f_{20}, z\}$

II SUBROUTINE STRES 3 (XE,C,PA,NCOLN,C,DU)

DIMENSION XE(20,3),C(60,NCOLN),DU(20,6,NCOLN)

XE,E,PA are the same parameters as defined earlier.

NCOLN - Number of columns in the force vector - the number of independent force systems assumed in the analysis.

C - Nodal displacements of an element (C(20x3,NCOLN) = C(60,NCOLN)) in the same order of nodal numbering.

DU - Stress values at the element nodes (in the same order of nodal numbering)

- The calculated six stresses at a node are three normal stresses $\sigma_x, \sigma_y, \sigma_z$ (in the global coordinate directions) and three shear stresses $\tau_{xy}, \tau_{yz}, \tau_{xz}$ for every independent force system.

Hence the matrix DU (20,6, NCOLN)
The program can be obtained with the following details:

KIND = PACK, PACKNAME = USER 3, TITLE = (U212S221) NEWISOCUBE