Use of constraint matrices in the finite element method and the stress analysis of complex rotating structures

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Additional Information:

• A Doctoral Thesis. Submitted in partial fulfillment of the requirements for the award of Doctor of Philosophy of Loughborough University.

Metadata Record: https://dspace.lboro.ac.uk/2134/14076

Publisher: © Susanta Kumar Roy

Please cite the published version.
This item was submitted to Loughborough University as a PhD thesis by the author and is made available in the Institutional Repository (https://dspace.lboro.ac.uk/) under the following Creative Commons Licence conditions.

**Creative Commons Licence**

**Attribution-NonCommercial-NoDerivs 2.5**

You are free:

- to copy, distribute, display, and perform the work

Under the following conditions:

**Attribution.** You must attribute the work in the manner specified by the author or licensor.

**Noncommercial.** You may not use this work for commercial purposes.

**No Derivative Works.** You may not alter, transform, or build upon this work.

- For any reuse or distribution, you must make clear to others the license terms of this work.
- Any of these conditions can be waived if you get permission from the copyright holder.

Your fair use and other rights are in no way affected by the above.

This is a human-readable summary of the Legal Code (the full license).

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
USE OF CONSTRAINT MATRICES IN THE
FINITE ELEMENT METHOD AND THE
STRESS ANALYSIS OF COMPLEX ROTATING STRUCTURES

by

SUSANTA KUMAR ROY, M.Sc.

A Doctoral Thesis submitted in partial fulfilment
of the requirements for the award of Doctor of Philosophy
of the Loughborough University of Technology

February 1973

Department of Mechanical Engineering

© by Susanta Kumar Roy, 1973
SUMMARY

The development of a method which allows different element types to be used in different parts of a complex structure with the help of equations of constraint has led to an economic means of finite element analysis of complex rotors by the most economic one of all the element types capable of representing the conditions in each segment reasonably well. One radial flow rotor, one curved blade doubly shrouded rotor, and one doubly shrouded rotor with hollow stiffened blades have been analysed using the technique called the "Constraint Matrix Method". Satisfactory results have been obtained in the first two cases and partially satisfactory results in the last case.

The method has been used in order to enforce inter-element displacement continuity in finite element formulation of structures and in the development of a triangular plate bending element stiffness matrix in which the initial nodes with the same number of degrees of freedom per node are replaced by a set of nodes with different number of degrees of freedom for different nodes necessary for achieving inter-element displacement continuity.

A three-dimensional solid finite element stiffness matrix has also been developed in terms of cylindrical coordinates based on the "mixed" method of analysis.
ACKNOWLEDGMENT

The writer wishes to express his gratitude to Mr. N.J. Cubitt for his guidance, valuable suggestions and critical comments throughout the planning and execution of this investigation.

He would also like to thank Professor Downs for his support and financial help, without which the experimental part of the work could not be carried out.

Thanks are also due to the Computer Centre of this University for their sympathy and help.

He also wishes to thank the technical staff of the Department of Mechanical Engineering, especially Mr G. Jones, Mr J. Burton, Mr P. Norton, and Mr S. Beet, for their expert services in the development of the test rig. Thanks are also due to Mr K. Topley for the excellent photographs of the test rig.

Above all, the writer would like to express his gratitude to Loughborough University of Technology for the grant which made this investigation possible.
NOMENCLATURE

Unless otherwise indicated the following symbols hold the assigned meanings throughout this thesis.

\( E \) - Young's Modulus

\( \nu \) - Poisson's Ratio

\( x, y, z \) - Cartesian coordinates

\( r, \theta, z \) - cylindrical coordinates

\( \sigma_x, \sigma_y, \sigma_z \) - direct stress components in Cartesian coordinates

\( \tau_{xy}, \tau_{yz}, \tau_{xz} \) - shear stress components in Cartesian coordinates

\( \sigma_r, \sigma_\theta, \sigma_z \) - direct stress components in cylindrical coordinates

\( \tau_{r\theta}, \tau_{rz}, \tau_{\theta r} \) - shear stress components in cylindrical coordinates

\( u, v, w \) - displacement components in Cartesian and cylindrical coordinates

\( \varepsilon_x, \varepsilon_y, \varepsilon_z \) - direct strain components in Cartesian coordinates

\( \gamma_{xy}, \gamma_{yz}, \gamma_{zx} \) - shear strain components in Cartesian coordinates

\( \varepsilon_r, \varepsilon_\theta, \varepsilon_z \) - direct strain components in cylindrical coordinates

\( \gamma_{r\theta}, \gamma_{rz}, \gamma_{\theta r} \) - shear strain components in cylindrical coordinates

\( D \) - flexural rigidity of a plate

\( G \) - modulus of rigidity

\( \omega \) - angular velocity in rad/sec
\( \rho \) - mass density
\( R \) - radius
\( RP \) - outer radius
\( g \) - unitless number equal to gravitational acceleration
\( \beta \) - \( \frac{\rho \times \omega^2}{g} \times Rp^2 \)
\( U \) - strain energy
\( U_T \) - total potential energy
\( \frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z} \) - partial differentials
\( \frac{\partial}{\partial r}, \frac{\partial}{\partial \theta}, \frac{\partial}{\partial \phi} \) -
\( v \) - \( \left( \frac{\partial}{\partial x} + \frac{\partial}{\partial y} + \frac{\partial}{\partial z} \right) \)
\( \ell, m, n \) - direction cosines
\{ \} - column vector or row vector
\( [ ]^T, [ ]^T \) - matrix transposition
\( [ ]^{-1} \) - matrix inversion
\( t \) - thickness
\( [C] \) - constraint matrix
## CONTENTS

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>2. SELECTION OF THE METHOD EMPLOYED</td>
<td>5</td>
</tr>
<tr>
<td>2.1 Previous Work</td>
<td>5</td>
</tr>
<tr>
<td>2.2 Possible Methods</td>
<td>8</td>
</tr>
<tr>
<td>2.2a The Method of Constraint Matrices</td>
<td>8</td>
</tr>
<tr>
<td>2.2b The Substructure and the Method of Influence Coefficients</td>
<td>9</td>
</tr>
<tr>
<td>2.2c Diakrptics</td>
<td>10</td>
</tr>
<tr>
<td>3. METHOD</td>
<td>13</td>
</tr>
<tr>
<td>3.1 Programming</td>
<td>22</td>
</tr>
<tr>
<td>4. APPLICATION OF THIS TECHNIQUE IN THE ANALYSIS OF ROTORS</td>
<td>24</td>
</tr>
<tr>
<td>4.1 Boundary Conditions</td>
<td>27</td>
</tr>
<tr>
<td>4.2 Junction of Shells</td>
<td>30</td>
</tr>
<tr>
<td>5. TESTS</td>
<td>31</td>
</tr>
<tr>
<td>5.1 Test Case 1</td>
<td>31</td>
</tr>
<tr>
<td>5.2 Test Case 2</td>
<td>34</td>
</tr>
<tr>
<td>6. THE ANALYSIS OF RADIAL FLOW IMPELLERS</td>
<td>37</td>
</tr>
<tr>
<td>7. THE ANALYSIS OF GENERAL CURVED VANE IMPELLERS</td>
<td>42</td>
</tr>
<tr>
<td>7.1 The Analysis of the Rotor Type BD1</td>
<td>42</td>
</tr>
<tr>
<td>7.1a Theoretical Analysis of the Rotor Type BD1</td>
<td>42</td>
</tr>
</tbody>
</table>
7.1b Experimental Verification of the Theoretical Results of Rotor Type BD1

7.2 The Analysis of the Rotor Type JH1

8. A THREE-DIMENSIONAL SOLID CYLINDRICAL FINITE ELEMENT

9. USE OF CONSTRAINT MATRICES IN THE DEVELOPMENT OF FINITE ELEMENT STIFFNESS MATRIX

10. DISCUSSION AND CONCLUSIONS

11. APPENDICES
   11.1 The Finite Element Method
       11.1a The Displacement Method
       11.1b The Mixed Method
   11.2 The Loughborough University Finite Element System
   11.3 The System used by the Writer
       11.3a Modifications made in the Existing Finite Element System Program
   11.4 Element Types used in this Report for the Analysis of Test Cases and Rotors
       11.4a The Plane Stress Element
       11.4b The Axisymmetric Solid Element
       11.4c The Shell Element
   11.5 Flow Charts
       11.5a Flow Chart of the Present System Program
       11.5b Flow Chart of the Program N309
   11.6 The Finite Element Meshes used in Chapters 6, 7, 8 and 9
   11.7 Graphs Presenting Results obtained in Chapters 6, 7 and 8
   11.8 The Test Rig for Strain Gauge Experiment on Rotor Type BD1

12. REFERENCES
1. INTRODUCTION

In 1970 a manufacturer of centrifugal impellers was considering the introduction of a range of impellers based on a few standard backplates and shrouds welded up with suitable vanes to give the mass flow and pressure characteristics required for a given application. The impellers thus produced would have been roughly similar to the one of the existing range shown in Figure 1.1.

The Department of Mechanical Engineering of this University, having been approached concerning the calculation of aerodynamic performance and stresses, prepared an estimate for the work which proved to be too high to be incorporated into the company's budget for the programme. It was, however, realised at the time that the stress analysis of such a rotor would require work of some novelty in the field of Finite Elements and the writer was asked to take up this work as a research project. It is now known that some finite element systems\textsuperscript{16,17,18} incorporate techniques rather similar in principle to those introduced by the writer, but there was at the time no published work on this or indeed on the stress analysis of any shrouded rotors with doubly curved vanes.

The only reliable method of stress analysis available to the manufacturer was the use of electrical resistance strain gauges, which eliminated the possibility of its use for fundamental design as would be required for the introduction of a complete new range. Such calculations as were carried out at the design stage were based on experience on
10 equally spaced doubly curved vanes

N.B.: DEF on vane is a free boundary

Figure 1.1: Shrouded Impeller

Axis of Rotation
service with earlier types of rotor and involved unsupported assumptions concerning the distribution of load between separate parts of the rotor.

This thesis will therefore seek to report first on the existing literature in connection with rotor analysis in Chapter 2.1.

Chapter 2.2 then explores a number of possible methods for solution of the current rotor problem and a critical assessment is then made, justifying the use of the technique actually used, namely the Finite Element Method with the application of Constraint Matrices to enable different types of elements to be combined and to facilitate the incorporation of the required boundary conditions. This method developed by the writer was presented at the last International Conference on Variational Methods.

Both the validity of the technique and the programming itself are tested for a few simple cases in which the results are known by other methods. These results are presented in Chapter 5, followed by a full analysis of radial and more general types of impellers in Chapters 6 and 7.

The work was carried out on the ICL 1904A computer at the Loughborough University. Facilities included up to 132K of core storage, together with two magnetic disc drive units and up to three magnetic tape units. There was available on this computer a Finite Element system (Chapter 11) with various types of element and a Cholesky solution routine for banded coefficient matrices. The elements used in the writer's work were:
(a) an axisymmetric three noded triangular element\textsuperscript{7} based on the displacement formulation;

(b) a plane stress three noded triangular element\textsuperscript{7} based on the displacement formulation;

(c) a polygonal flat-plate shell element\textsuperscript{15} with seven degrees of freedom per node based on the mixed formulation.

These elements are described later in more detail (Chapter 11.4) but are mentioned now in order to define the starting point of the writer's work. In fact the writer had to do a considerable amount of work on the system and the element (c) above in order to make them work satisfactorily. Some of this work is described in Chapter 11.3a.

No fully three-dimensional element was available in the existing system and none was essential for the analysis of curved vane shrouded rotors dealt with by the writer. However the need for such elements in the analysis of many rotors is obvious, and so the writer has done further work on the development of a cylindrical polar three-dimensional element of mixed formulation. This is described in Chapter 8.

Finally the writer demonstrates the use of a transformation matrix (this is a particular type of the constraint matrix) in the formulation of an element stiffness matrix, enabling the use of a uniform number of degrees of freedom per node at the stage of formulation although finally different nodes may possess different number of degrees of freedom, and enforcing compatibility along inter-element boundaries. This is illustrated in Chapter 9 by consideration of a plate bending element.
2. SELECTION OF THE METHOD EMPLOYED

2.1 Previous Work

The analysis of rotating bodies by the classical theories of elasticity\(^1\) has been confined to relatively simple problems, where the bodies are axially symmetric and the profile is given by a simple geometric expression. The stresses are usually assumed to remain constant across the thickness although there are solutions\(^1\) available for bodies of uniform thickness with stresses varying across the thickness.

There have been several attempts\(^2,3\) to analyse discs where the profile has no plane of symmetry, by the approximate solution of governing differential equations. The method used is generally one of the following two:

In the first, known as the finite difference method, the governing differential equations are discretised in terms of finite steps of the domain. This leads to a set of linear equations to solve for an approximate answer. The accuracy of the results depends upon the step size chosen and numerical errors introduced due to rounding off. The problem with this method is that the governing differential equations have to be formulated first and then they have to undergo a process of discretisation which becomes more difficult for higher order equations. The process of imposition of boundary conditions is also complicated in this method.

In the second, known as the point matching method, a finite number of terms are taken from an infinite series...
solution of the governing differential equations. Then enforcing this finite series to have the prescribed boundary values at several points, the coefficients of the terms in the series are found. The accuracy of this method is dependent on the length of the finite series solution chosen to replace the infinite series. However, it is often impossibly difficult to formulate the governing differential equations and find a series solution for them. Moreover, the resulting linear equations tend to suffer from ill-conditioning effect with the use of close boundary points.

When vanes are added to a disc, the problem becomes even more complicated and the techniques involving approximation of exact differential equations become even more difficult. Although some finite difference solutions of rotors with radial vanes are available, this method becomes too complex for use in the analysis of more general rotors.

However, the finite element method, because of its versatility and adaptability, can be successfully applied to the analysis of such rotors.

The analysis of a rotor by the finite element method can, in principle, be accomplished by treating it as a fully three-dimensional body composed of three-dimensional solid elements. But because of complicated geometry and stress patterns, a very large number of sophisticated three-dimensional elements are required for a satisfactory solution. In particular it can be seen that, depending on the type of three-dimensional solid element chosen, the use of such elements for thin shells requires the use of many more
elements than if shell elements are used simply in order to avoid ill-conditioning effect associated with odd-shaped elements and to get a convergent solution\(^2\). Moreover, because of the complicated geometry, the effort required for data preparation may be excessive. Although McEwan and Hellen\(^9\) have recently presented a fully three-dimensional analysis of small curved-blade impellers, Chan and Henrywood\(^6\) have reported excessive data preparation and computation time required for such analysis. Thus the human effort and computer time required for this sort of analysis of large rotors compels one to look for alternative approaches.

A finite element analysis of axisymmetric rotors has been presented by Stordahl and Christensen\(^5,8\), but as the problem has been treated as fully axisymmetric, the analysis is not suitable for more general rotors.

Chan and Henrywood\(^6\) have presented a finite element analysis of rotors with radial vanes using a new three-dimensional element for the axisymmetric part, designed to be compatible with plane stress elements used in the blades at the interface. However, this analysis is still limited to rotors with radial vanes only.

It appears that the only suitable alternative to using fully three-dimensional elements in all parts of an impeller, is to divide the impeller into several parts and treat each part with the most suitable element type considering the required accuracy of the results, the effort available for data preparation and the computer time available on the budget for the project.

* This part, though geometrically axisymmetric, is not loaded truly axisymmetrically and thus has a three-dimensional stress distribution.
2.2 Possible Methods

One should now consider the possible ways of handling the above approach to the analysis:

2.2a The Method of Constraint Matrices

A method is proposed here for the analysis of rotors involving the equations of constraint\textsuperscript{11,25}.

Suppose that a structure is divided into two segments with \([K_1]\) and \([K_2]\) as their individual stiffness matrices. Let \({u_1}\) and \({u_2}\) be the generalised displacements of the two segments respectively.

Now, if the displacements \({u_1}\) and \({u_2}\) can be written in terms of an overall generalised displacement vector \({q}\) and if we postulate the existence of an overall stiffness matrix, \([K]\), relating the overall generalised loads with \({q}\), then

\[
\begin{bmatrix}
{u_1} \\
{u_2}
\end{bmatrix} = [C] [q] \quad \text{............... (2.1)}
\]

and

\[
[K] = [C]^t \begin{bmatrix}
[K_1] & [0] \\
[0] & [K_2]
\end{bmatrix} [C] \quad \text{............... (2.2)}
\]

If \({F_1}\) and \({F_2}\) are individual load vectors and \({F}\) is the overall load vector then

\[
{F} = [C]^t \begin{bmatrix}
{F_1} \\
{F_2}
\end{bmatrix} \quad \text{............... (2.3)}
\]

Thus the segments are now integrated into a combined system.
This approach is simple, easy to handle, and easy to program. The human effort required in data preparation is small for this operation and the computer time used in handling the constraints is likely to be a small part of the total time of analysis. Other advantages of this method include the capability of handling the whole structure at once and the ease with which the boundary conditions can be incorporated.

A detailed description of this method is given in Chapter 3.

2.2b The Substructure and the Method of Influence Coefficients

Substructures are used in the finite element analysis of large structures. A structure is divided into several substructures and each substructure is divided into a large number of finite elements. The external nodes which correspond to some nodes of another substructure are called supernodes. Each substructure is analysed first. Forces and displacements due to unit displacements of supernodes are determined and a stiffness matrix for a substructure is formed. Forces and displacements due to external loads are also calculated keeping supernodes fixed. Taking each substructure as one element, a system analysis is carried out for the whole structure.

In the case of impellers, each section of the rotor, formulated by one type of element only, may be taken as a substructure. But in this case the final assembly of the substructure is not straightforward because of lack of
identity and direct correspondence between the adjoining nodes of two substructures.

However, if adequate equilibrium equations can be set up relating forces at the supernodes of one substructure and the corresponding supernodes on an adjoining substructure, the forces at the supernodes of the adjoining substructure due to unit displacements at the supernodes of one substructure can be determined. Hence the influence coefficients relating the two substructures can be found and thus an overall solution can be obtained after choosing the appropriate generalised coordinates.

But this process is likely to be extremely laborious, difficult to translate into a computer program, and costly in terms of computer time. Except for extremely large structures this method does not have any advantages over the constraint matrix method discussed before, provided that the combined stiffness matrix can be handled by the computer.

2.2c Diakoptics

This is a method of handling large systems by tearing the system into several segments. This method has been predominantly used by electrical engineers for the analysis of large networks.

In simple form the Diakoptical method adapted in the context of finite element analysis of complex structures may be given as follows:

Suppose that a structure has been divided into two segments, (1) and (2), $[K_1]$ and $[K_2]$ are their individual
stiffness matrices, \{u_1\} and \{u_2\} are their individual generalised displacements, and \{F_1\} and \{F_2\} are their individual external load vectors.

Let \{F_1'\} and \{F_2'\} be the load vectors imposed in each section due to segmentation.

Then
\[
\begin{bmatrix}
[K_{11}] & [0] \\
[0] & [K_{22}]
\end{bmatrix}
\begin{bmatrix}
\{u_1\} \\
\{u_2\}
\end{bmatrix} =
\begin{bmatrix}
\{F_1\} \\
\{F_2\}
\end{bmatrix} +
\begin{bmatrix}
\{F_1'\} \\
\{F_2'\}
\end{bmatrix}
\] ....(2.4)

Now we assume the existence of equations of equilibrium such that
\[
\{F'_\alpha\} = [C_{\alpha\psi}]{F_{\psi}}
\] ...............(2.5)

where \{F_{\psi}\} are the unknown nodal forces at the common nodes in the combined system.

If \{q_{\psi}\}, the generalised displacements of common nodes in the combined system are given by
\[
\{q_{\psi}\} = - [C'_{\psi\alpha}]{u_\alpha}
\] ...............(2.6)

and if a relationship between \{q_{\psi}\} and \{F_{\psi}\} exists such that
\[
\{q_{\psi}\} = [Z_{\psi\psi}]{F_{\psi}}
\] ...............(2.7)

then from eqn (2.4)
\[
\begin{bmatrix}
[K_{11}] & [0] & [C'_{1\psi}] \\
[0] & [K_{22}] & [C'_{2\psi}] \\
[C_{\psi1}] & [C_{\psi2}] & [Z_{\psi\psi}]
\end{bmatrix}
\begin{bmatrix}
\{u_1\} \\
\{u_2\} \\
\{F_{\psi}\}
\end{bmatrix} =
\begin{bmatrix}
\{F_1\} \\
\{F_2\} \\
\{0\}
\end{bmatrix}
\]
Then by elimination

\[
\{u_α\} = \{K′_αα\}^{-1} - \{K'_αα\}^{-1} \{C_αψ\} \{Z''_ψψ\}^{-1} \{C'_αψ\} \{K_αα\} \{F_α\} \quad \cdots \cdots (2.8)
\]

where

\[
\{Z''_ψψ\} = \{Z'_ψψ\} + \{C'_αψ\} \{K_αα\}^{-1} \{C_αψ\}.
\]

An electrical analogy of the above process and the above equation can be found in reference number 14.

This method is designed for very large systems where solution of the equations cannot be found directly.

However in the context of finite element analysis of rotors, it is not very convenient. It is tedious and time consuming. Moreover some basic problems may arise, for example, \(\{K_αα\}\) may be found to be singular.

Thus none of the above two methods has any advantage over the direct constraint matrix method for problems, such as fair size rotors, in which this method can be used. So for its simplicity and ease of handling, the constraint matrix method was chosen for the purpose of this investigation.
3. **METHOD**

It is proposed here to derive a method enabling the use of different element formulations for different parts of a complex structure for a unique finite element solution, using an established structural technique.

Some different types of elements can be used together when the generalised coordinates of the element with less degrees of freedom are the same as some of those of the other elements, and the elements of one type meet the elements of the other type at the common nodes. Consider an element type in which the generalised coordinates are \( p, q, r \) and \( s \). Thus \( [K](u) = \{F\} \) where \([K]\) is the element stiffness matrix; \( \{u\} \) are the displacement components of \( p, q, r \) and \( s \); and \( \{F\} \) is the nodal force vector. Another type of element is present in which the generalised coordinates are \( p, q \). Then \( [K_i](u_i) = \{F_i\} \) where \( i \) represents the second type of element.

To use the above types of elements together it is sufficient to increase the size of \([K_i]\) and \([F_i]\) by filling the elements corresponding to the generalised coordinates \( r \) and \( s \) with zeroes. Then these two element stiffness matrices may be combined by adding the stiffness contributions to a node by connecting elements. The force vector can be dealt with similarly. The continuity of displacement along common boundaries between these two element types may or may not be broken. If it is broken some localised disturbances will occur. As an example, this technique can be used for a structure consisting of a plate resting on beams.
Considering Figure 3.1, the node B on the beam may have 2 degrees of freedom $(w, \theta x)$, so that the segment of an element stiffness matrix corresponding to this node may be given by

$$ [K_B] = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \quad \ldots \quad (3.1) $$

The node P on the plate may have 3 degrees of freedom $(w, \theta x, \theta y)$, so that the segment of an element stiffness matrix corresponding to this node may be given by
The equation (3.1) may be written as
\[
\begin{bmatrix}
K_{11} & K_{12} & K_{13} \\
K_{21} & K_{22} & K_{23} \\
K_{31} & K_{32} & K_{33}
\end{bmatrix}
\]

The equation (3.1) may be written as
\[
\begin{bmatrix}
K_{11} & K_{12} & 0 \\
K_{21} & K_{22} & 0 \\
0 & 0 & 0
\end{bmatrix}
\] (3.3)

Now the node 1 on the beam may be considered to have 3 degrees of freedom (w, $\theta x$, $\theta y$) and its stiffness contributions to node 2 may then be added directly to those from plate elements given by equation (3.2).

However the application of the above technique is limited and a more generally applicable method is now described. Suppose that a structure is divided into two parts with forces applied at the junction to equalise displacements in two parts at the junction. Let the part 1 be described by a set of generalised displacements $\{u_1\}$ and the part 2 by a set of generalised displacements $\{u_2\}$. Then
\[
\{K_1\} \{u_1\} = \{F_1\}
\]
and
\[
\{K_2\} \{u_2\} = \{F_2\}
\]

The concept is now introduced of constraint matrices $[C_1]$ and $[C_2]$ which relate the generalised displacement for the two separate parts to those for the combined structure. If the generalised displacements for the integrated structure are given, $\{u'\}$, then $\{u_1\}$ and $\{u_2\}$ can be given by
\[ \{u_1\} = [C_1]\{u'\} \quad \ldots \ldots \quad (3.6) \]

and
\[ \{u_2\} = [C_2]\{u'\} \quad \ldots \ldots \quad (3.7) \]

As the combined system is statically identical to the two separate systems the strain energies must be equal.

The strain energy contained in the combined system is given by
\[ U_T = \frac{1}{2}\{u_1\}^T[K_1]\{u_1\} + \frac{1}{2}\{u_2\}^T[K_2]\{u_2\} \]
\[ = \frac{1}{2}\{u'\}^T[C_1]^T[K_1][C_1]\{u'\} + \frac{1}{2}\{u'\}^T[C_2]^T[K_2][C_2]\{u'\} \]
\[ = \frac{1}{2}\{u'\}^T[C]^T[K][C]\{u'\} \]
\[ = \frac{1}{2}\{u'\}^T[K']\{u'\} \quad \ldots \ldots \quad (3.8) \]

where
\[ [C] = \begin{bmatrix} [C_1] \\ [C_2] \end{bmatrix} \quad \ldots \ldots \quad (3.9) \]

and
\[ [K] = \begin{bmatrix} [K_1] & [0] \\ [0] & [K_2] \end{bmatrix} \quad \ldots \ldots \quad (3.10) \]

therefore
\[ [K'] = [C]^T[K][C] \quad \ldots \ldots \quad (3.11) \]

Similarly external work done by the loading must be the same whether the systems are combined or not. Thus
\[ -\{u'\}^T[F'] = -\{u_1\}^T[F_1] - \{u_2\}^T[F_2] \]
therefore
\[ (u')^T(F') = (u')^T[C_1]^T(F_1) + (u')^T[C_2]^T(F_2) \]
\[ = (u')^T[C_1]^T[F_1]^T[C_2]^T[F_2] \]
\[ = (u')^T[C]^T(F) \] ......... (3.12)
and
\[ (F') = [C]^T(F) \] ......... (3.13)
where
\[ (F) = \begin{bmatrix} (F_1) \\ (F_2) \end{bmatrix} \] ......... (3.14)

Thus from \([K_1], [K_2]\) and \([C]\) it is possible to obtain \([K]\), the stiffness matrix of the combined system, and from \((F)\) and \([C]\), the combined load vector, \((F')\), is obtained. So it is possible to treat a part of a structure separately from another part and to combine them for a unique solution provided the equations describing relationship between the generalised coordinates chosen to analyse one part and those for the other part can be established.

It should be noted that the unknown forces at the junction of the two parts are always eliminated in the process just described. True applied forces at the junction may be treated as applied to one part or the other, or divided between them. It should be remembered that if displacements are not continuous along the boundary between the two parts, some localised disturbances will occur.
As an illustration consider the following simple system:

![Image of a simple system](image)

Figure 3.2

The equations of equilibrium are as follows:

\[
\begin{align*}
K_3 u_1' - K_4 (u_2' - u_1') &= 0 \\
K_4 (u_2' - u_1') + K_1 (u_2' - L_1 \theta') + K_2 (u_2' + L_2 \theta') &= P' \\
-K_1 (u_2' - L_1 \theta') \times L_1 + K_2 (u_2' + L_2 \theta') \times L_2 &= 0
\end{align*}
\]

Thus

\[
\begin{bmatrix}
(K_3 + K_4) & -K_4 & 0 \\
-K_4 & (K_1 + K_2 + K_4) & (K_2 L_2 - K_1 L_1) \\
0 & (K_2 L_2 - K_1 L_1) & (K_2 L_2^2 + K_1 L_1^2)
\end{bmatrix}
\begin{bmatrix}
u_1' \\
u_2' \\
\theta'
\end{bmatrix}
= 
\begin{bmatrix}
0 \\
P' \\
0
\end{bmatrix}
\] (3.15) (3.16)

Now consider the following systems:
For the system in Figure 3.3a the equilibrium equations are

\[
\begin{align*}
K_3u_1 - K_4(u_2 - u_1) &= 0 \\
K_4(u_2 - u_1) &= P_2 + P'
\end{align*}
\]  
................ (3.17)

So that

\[
\begin{bmatrix}
K_3 + K_4 & -K_4 \\
-K_4 & K_4
\end{bmatrix}
\begin{bmatrix}
\{u_1\} \\
\{u_2\}
\end{bmatrix} = 
\begin{bmatrix}
\{Q\} \\
\{P' + P_2\}
\end{bmatrix}
\]  
................ (3.18)

Similarly for systems in Figure 3.3b and Figure 3.3c respectively

\[
\begin{align*}
K_1u_3 &= P_3 \\
K_2u_4 &= P_4
\end{align*}
\]  
................ (3.19)

If these systems are to be assembled into the system
shown in Figure 3.2 then the equations of constraint are
\[
\begin{align*}
    u_1 &= u_1' \\
    u_2 &= u_2' \\
    u_3 &= u_2' - L_1 \theta' \\
    u_4 &= u_2' + L_2 \theta'
\end{align*}
\]
\[...............(3.21)\]

Thus
\[
\begin{pmatrix}
    u_1 \\
    u_2 \\
    u_3 \\
    u_4
\end{pmatrix} =
\begin{pmatrix}
    1 & 0 & 0 & 0 \\
    0 & 1 & 0 & 0 \\
    0 & 1 & -L_1 & 0 \\
    0 & 1 & L_2 & 0
\end{pmatrix}
\begin{pmatrix}
    u_1' \\
    u_2' \\
    \theta'
\end{pmatrix}
\]
\[...............(3.22)\]

The constraint matrix \([C]\) is as follows:
\[
[C] =
\begin{pmatrix}
    1 & 0 & 0 & 0 \\
    0 & 1 & 0 & 0 \\
    0 & 1 & -L_1 & 0 \\
    0 & 1 & L_2 & 0
\end{pmatrix}
\]
\[...............(3.23)\]

Also if static equilibrium is to be maintained in the combined system
\[
P_2 = -(P_3 + P_4)
\]
\[...............(3.24)\]

Now using the constraint matrix method the stiffness matrix of the combined system is given by
\[
[K'] =
\begin{pmatrix}
    1 & 0 & 0 & 0 \\
    0 & 1 & 1 & 1 \\
    0 & 0 & -L_1 & L_2
\end{pmatrix}
\begin{pmatrix}
    (K_3+K_4) & -K_4 & 0 & 0 \\
    -K_4 & K_4 & 0 & 0 \\
    0 & 0 & K_1 & 0 \\
    0 & 0 & 0 & K_2
\end{pmatrix}
\begin{pmatrix}
    1 & 0 & 0 \\
    0 & 1 & 0 \\
    0 & 1 & -L_1 \\
    0 & 1 & L_2
\end{pmatrix}
\]
Similarly the load vector is given by

\[
\{F'\} = \begin{bmatrix}
1 & 0 & 0 & 0 \\
0 & 1 & 1 & 1 \\
0 & 0 & -L_1 & L_2
\end{bmatrix} \begin{bmatrix}
P'_1 \\
P_2 \\
P_3 \\
P_4
\end{bmatrix} = \begin{bmatrix}
P'_1 + P_2 \\
(P_2 + P_3 + P_4 + P'_1) \\
(-L_1 P_3 + L_2 P_4)
\end{bmatrix}
\]

Substituting from equation (3.24)

\[
P_2 + P_3 + P_4 = 0
\]

Also for overall equilibrium

\[
-L_1 P_3 + L_2 P_4 = 0
\]

Thus it can be seen that the dummy forces introduced in the broken systems to give the same nodal displacements as those in the original system are automatically eliminated and need not enter into computation.

Now from above

\[
\begin{bmatrix}
(K_3+K_4) & -K_4 & 0 \\
-K_4 & (K_1+K_2+K_3) & (K_2 L_2 - K_1 L_1) \\
0 & (K_2 L_2 - K_1 L_1) & (L_1^2 K_1 + L_2^2 K_2)
\end{bmatrix} \begin{bmatrix}
u'_1 \\
u'_2 \\
\theta
\end{bmatrix} = \begin{bmatrix}
0 \\
P' \\
0
\end{bmatrix}
\]

\[\ldots(3.29)\]
The identity of the equations (3.16) and (3.29) illustrates the method.

3.1 Programming

By suitable partitioning of the matrices and by using high speed read-write facilities, the above operations can be accomplished in a computer, without excessive demands on time or storage.

$[C]$ is in general of the form

\[
[C] = \begin{bmatrix} [I] & [0] & [0] \\ [0] & [C] & [0] \\ [0] & [0] & [I] \end{bmatrix}
\]

............... (3.30)

where $[I]$ is the identity matrix.

$[K]$ can be partitioned as

\[
\]

............... (3.31)

Then

\[
\]

............... (3.32)

Because of symmetry of $[K]$ and $[C]^T[K][C]$ some of the above multiplications are not necessary.

When $\{u'\}$ has been obtained, the original displacement vector $\{u\}$ can be obtained from
\{u\} = [C] \{u'\}

\[
\begin{bmatrix}
[1] & [0] & [0] \\
[0] & [C] & [0] \\
[0] & [0] & [1]
\end{bmatrix}
\begin{bmatrix}
\{B_1\} \\
\{B_2\} \\
\{B_3\}
\end{bmatrix}
\]

\[
\begin{bmatrix}
\{B_1\} \\
\{C_t\} \{B_2\} \\
\{B_3\}
\end{bmatrix}
\]

\(\text{...............(3.33)}\)

A detailed description of the programs (N309 and N312) to perform the above operations is given in Chapter 11.
4. APPLICATION OF THIS TECHNIQUE IN THE ANALYSIS OF ROTORS

The method used here for the analysis of the general rotor was to divide the structure into a number of parts, with shell elements used in the vanes and other parts thin enough to be treated as thin shells, and with appropriate types of elements for other sections.

When stiffness matrices and load vectors of these different parts have been developed, the overall stiffness matrix and the load vector for the integrated structure is obtained by first establishing algebraic equations relating displacements of some nodes of one part with displacement of some nodes of another part, and then forming the constraint matrix, \([C]\), from these equations. This way it is possible to analyse most general rotor problems comparatively inexpensively.

4 identical vanes

Part B (one of four identical quadrants)

Axis of rotation

Figure 4.1
For example the hypothetical rotor shown in Figure 4.1 may be divided into two parts. Let the part B be described by shell\textsuperscript{15} and the part A by axisymmetric\textsuperscript{7} elements.

If \( u, v \) and \( w \) are three displacement components of a shell node in Cartesian coordinates and \( u_r \) and \( u_T \) are radial and tangential displacements of shell nodes on the boundary with the axisymmetric part,

\[
\begin{align*}
    u_r &= w \cos \theta + u \sin \theta \\
    u_T &= u \cos \theta - w \sin \theta \\
    \text{therefore} \\
    u &= u_r \sin \theta + u_T \cos \theta \\
    w &= u_r \cos \theta - u_T \sin \theta 
\end{align*}
\]

Similarly rotations are given by

\[
\begin{align*}
    \theta_x &= \theta_r \sin \theta + \theta_T \cos \theta \\
    \theta_z &= \theta_r \cos \theta - \theta_T \sin \theta
\end{align*}
\]

Axisymmetric nodes have two degrees of freedom \((u,v)\) and shell nodes have seven degrees of freedom \((u,v,\theta_x,\theta_y, \theta_z,e)\). \( e \) is dilation and given by

\[
e = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}
\]

The displacement components of shell nodes at the boundary with the axisymmetric part are related to the displacements of axisymmetric nodes as follows:
The below equations can be presented in matrix form as:

\[
\begin{align*}
\mathbf{u} &= \frac{u_K + u_M}{2} \times \sin \theta \\
\mathbf{v} &= \frac{v_K + v_M}{2} \\
\mathbf{w} &= \frac{u_K + u_M}{2} \times \cos \theta \\
\mathbf{\theta x} &= -\frac{u_K - u_M}{KM} \times \cos \theta \\
\mathbf{\theta y} &= 0 \\
\mathbf{\theta z} &= \frac{u_K - u_M}{KM} \times \sin \theta \\
\mathbf{e} &= \left(\frac{u_K + u_M}{2}\right) \times \frac{1}{R} + \left(\frac{u_K + u_M}{2} - \frac{u_I + u_J}{2}\right) \beta \\
&\quad + \left(\frac{v_M - v_K}{KM}\right)
\end{align*}
\]

The above matrix can be readily incorporated into a...
computer program for the generation of $[C]$. Then this $[C]$ matrix can be used as described before for an overall solution.

4.1 Boundary Conditions

For a rotor with several identical vanes, the rotor can be divided into as many identical segments as there are vanes. It is necessary to consider one such segment only, but for a rotor with curved vanes it becomes difficult to establish the boundary conditions explicitly. However it is possible to form some algebraic equations relating displacements in cylindrical coordinates, at one edge of a segment with the corresponding displacements on the other edge of the segment. From these algebraic equations a constraint matrix can be developed and can be used to reduce the stiffness matrix and the force vector leading to the solution in the way described above.

Suppose nodes 1 and 2 are at a particular radius on two opposite and similar boundaries (shown in Figure 4.2) of the segment under consideration. Then in cylindrical coordinates

Figure 4.2
\[ u_{r1} = u_{r2}; \quad u_{t1} = u_{t2}; \quad \theta_{r1} = \theta_{r2}; \quad \theta_{t1} = \theta_{t2} \quad \ldots (4.5) \]

where \( r \) and \( t \) represent radial and tangential directions respectively.

From equations (4.1), (4.2), and (4.5)

\[
\begin{align*}
\sin \theta_1 + w_1 \cos \theta_1 &= u_2 \sin \theta_2 + w_2 \cos \theta_2 \\
\cos \theta_1 - w_1 \sin \theta_1 &= u_2 \cos \theta_2 - w_2 \sin \theta_2 \\
\theta_{x1} \sin \theta_1 + \theta_{z1} \cos \theta_1 &= \theta_{x2} \sin \theta_2 + \theta_{z2} \cos \theta_2 \\
\theta_{x1} \cos \theta_1 - \theta_{z1} \sin \theta_1 &= \theta_{x2} \cos \theta_2 - \theta_{z2} \sin \theta_2 \\
v_1 &= v_2 \\
e_1 &= e_2
\end{align*}
\]

(4.6)

where \( u, v, w, \theta_x, \theta_y, \theta_z \) refer to the cartesian coordinates.

Then

\[
\begin{align*}
u_2 &= u_1(\sin \theta_1 \sin \theta_2 + \cos \theta_1 \cos \theta_2) + w_1(\cos \theta_1 \sin \theta_2 - \sin \theta_1 \cos \theta_2) \\
w_2 &= u_1(\sin \theta_1 \cos \theta_2 - \cos \theta_1 \sin \theta_2) + w_1(\cos \theta_1 \cos \theta_2 + \sin \theta_1 \sin \theta_2)
\end{align*}
\]

(4.7)

Similarly for \( \theta_{x2} \) and \( \theta_{z2} \).

Thus the displacements of node 2 are related to the displacements of node 1.

Now the above equations can be presented in matrix form as below:
\[
\begin{align*}
\begin{bmatrix}
u_1 \\
v_2 \\
w_1 \\
\theta_{x1} \\
\theta_{y1} \\
\theta_{z1} \\
e_1 \\
u_2 \\
v_2 \\
w_2 \\
\theta_{x2} \\
\theta_{y2} \\
\theta_{z2} \\
e_2
\end{bmatrix}
= \begin{bmatrix}
1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 \\
\frac{\sin \theta_1 \sin \theta_2 + \cos \theta_1 \cos \theta_2}{\sin \theta_1 \sin \theta_2 - \cos \theta_1 \cos \theta_2} & \frac{\cos \theta_1 \sin \theta_2}{-\sin \theta_1 \cos \theta_2} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\
\frac{\sin \theta_1 \cos \theta_2 + \cos \theta_1 \sin \theta_2}{\sin \theta_1 \cos \theta_2 - \cos \theta_1 \sin \theta_2} & \frac{\cos \theta_1 \cos \theta_2}{-\sin \theta_1 \sin \theta_2} & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & \frac{\sin \theta_1 \cos \theta_2}{-\cos \theta_1 \sin \theta_2} & \frac{\cos \theta_1 \cos \theta_2}{\sin \theta_1 \sin \theta_2} & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\begin{bmatrix}
u_1 \\
v_1 \\
w_1 \\
\theta_{x1} \\
\theta_{y1} \\
\theta_{z1} \\
e_1
\end{bmatrix}
\end{align*}
\]
Again, this matrix can be readily incorporated into a computer program for the generation of $[C]$ which can be used as described before for an overall solution.

4.2 Junctions of Shells

The stiffness matrices for the shell elements are developed first in terms of a local coordinate system and then transferred to the global coordinate system by means of a transformation matrix. Therefore if the shell elements used in different parts meeting at a junction have the same generalised degrees of freedom per node in the global coordinate system, then assembly of these elements presents no problem.

However if different types of shell elements are used for different parts meeting at a junction giving rise to mutually incompatible element stiffness matrices at the junction, then the constraint matrix method can be used for their assembly provided algebraic equations can be set up relating the generalised coordinates of all individual nodes on different parts at the junction to a set of overall generalised coordinates. Sometimes it may be necessary to use high-order three-dimensional solid elements near a junction for better accuracy while treating the portions away from the junction by ordinary shell elements. In this case and other similar cases the constraint matrix method can be used for an overall solution, but the matter of discontinuity of displacements along the joining lines of different parts in these cases must be carefully considered.
5. TESTS

In order to establish the validity of the technique a number of tests have been carried out. The results obtained in two of these tests are given below. As it is the displacement which is specified in the formulation of the technique, the results are presented in terms of displacements.

5.1 Test Case 1

In this case the problem analysed is a circular disc of uniform thickness (0.2") and of radius 1.0".

The disc is subjected to two different loading cases - in load case (a) a uniformly distributed load of total value 2400 lbf acts radially along the periphery and across the thickness. In load case (b), in addition to the load acting in case (a), a load of 2400 lbf is distributed uniformly axially along the periphery as shown in Figure 5.1a.

Figure 5.1a
These are analysed first by axisymmetric finite element formulation by the displacement approach using the constant strain element described in Chapter 11.4b. The finite element mesh is uniform in this case and is made by eight divisions through the thickness and twenty divisions in the radial direction. The results, displacements of the points A and B shown in Figure 5.1b, are identified by CASE1a for load case (a) and CASE1b for load case (b).

Then it is divided into two sections shown in Figure 5.1b. Section (1) is formulated by axisymmetric elements as in the previous approach with an identical mesh, and Section (2) is formulated by shell elements, as shown in Figure 5.1b, (Chapter 11.4c). Then the two sections are combined by a constraint matrix. The boundary conditions for nodes 6 and 10 are introduced by a constraint matrix, generated by the computer from node numbers and nodal...
coordinates, by relating displacements of node 10 with those of node 6 in terms of cylindrical coordinates. The results for this case are identified by CASE2a for load case (a) and CASE2b for load case (b).

The results are presented in Table 5.1 for comparison between the two approaches.

<table>
<thead>
<tr>
<th>Points</th>
<th>DISPLACEMENTS (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>CASE 1a</td>
</tr>
<tr>
<td></td>
<td>Radial</td>
</tr>
<tr>
<td>A</td>
<td>0.00004</td>
</tr>
<tr>
<td>B</td>
<td>0.00005</td>
</tr>
</tbody>
</table>

From the results presented above it can be seen that there is good correlation between the results obtained in CASE1a and CASE2a as well as between CASE1b and CASE2b.
5.2 TEST CASE 2

This is a simple plane stress problem of a plate under pure tension analysed by triangular finite elements with a linear displacement field. The plate is square (4' x 4'). A quarter of the plate is used as shown in Figure 5.2a.

![Figure 5.2a](image1)

**Case 2a**

Here this simple problem has been solved by the finite element method with a mesh as shown in Figure 5.2a.

**Case 2b**

Here the problem is solved by the finite element method with a mesh as shown in Figure 5.2b. Node 5a has no contribution from element (4), so that there is no guarantee of compatibility along the line 5-6.

**Case 2c**

Here the problem is solved by the same mesh as in Figure 5.2b, but this time a constraint matrix is used to enforce compatibility along the line 5-6, by stating that the displacements of node 5a are half the sum of the corresponding displacements of nodes 5-6.

The results obtained are given in Table 5.2.
Table 5.2

From the results above the displacements of nodes in case 2b are not very conspicuously different from those in Case 2c. But assuming a linear distribution of displacements between nodes, it can be seen from the displacements of nodes 5, 5a, and 6 in Case 2b, that the node 5a does violate compatibility, whereas in Case 2c node 5a preserves compatibility.

Although displacements of most nodes do not show any marked influence of node 5a either in Case 2b, or Case 2c, the displacements of nodes 8 and 9 in Case 2c are much closer to the values obtained in Case 2a than those obtained in Case 2b.
Thus from this it can be concluded that constraint matrices can be used to enforce compatibility and thereby to obtain better results.
6. THE ANALYSIS OF RADIAL FLOW IMPELLERS

Although the objective of the project was primarily to develop an economic method of stress analysis for the rotor type shown in Figure 1.1, as a first approach it was decided to analyse a radial flow impeller, for which other theoretical results are available, in order to compare the results obtained by the method used by the writer and those obtained by the other investigators.

Figure 6.1 shows a typical radial flow impeller which has been analysed theoretically and experimentally by various other investigators by the finite difference method, the finite element method and the photoelastic method. The results obtained by the above methods are given in Reference 6. The results obtained here are compared with those given in the above reference for a centrifugal loading system.

For the purpose of analysis by the present method, the impeller has been divided into three parts:

1. the lower part of the back plate, up to the line A-A shown in Figure 6.1, is considered to have axially symmetric stress and consequently treated by axisymmetric solid elements (see Chapter 11.4b);
2. the thin upper part of the back plate, above the line A-A shown in Figure 6.1 is assumed to behave as a shell and is treated by flat plate shell elements (see Chapter 11.4c);
3. the blades are treated by shell elements although they can be adequately represented by plane stress
Figure 6.1: Radial Impeller (17 blades)
elements. This has been done as a prelude to the analysis of rotor types, shown in Figure 1.1, where the vanes behave as shells.

The system was analysed with the coarse mesh shown in Figure 6.2 (see Chapter 11.6) with ninety effective degrees of freedom. The material constants assumed were \( E = 10^7 \text{lbf/in}^2 \) and \( v = 0.34 \) as used by the other investigators. Because of symmetry only one of the seventeen identical segments of the backplate, each with a blade, was analysed. In this case the following constraining operations were necessary for an overall solution:

1. to combine the axisymmetric part of the backplate with the shell part by matching the displacements of nodes k, l and m on the shell part with those of the nodes g, h, i and j on the axisymmetric part (see Chapter 4);

2. to impose boundary conditions by equating displacements of node n to those of node p in terms of cylindrical coordinates (see Chapter 4);

3. to combine the blade with the axisymmetric part by equating displacements of nodes d, e and f to those of nodes a, b and c respectively. (Note that the shell part of the backplate and the blade are treated by the same type of element and therefore they are combined automatically because of common nodes.)

The results obtained are presented in graphs 6.1 to 6.6 (see Chapter 11.7) in comparative form with results obtained by other investigators. In the assumed axisymmetric part of
the backplate, the calculated stresses were assumed to be acting beneath the blades, although the analysis of Chan and Henrywood showed very little difference between stresses beneath the blades and those between the blades at the rear of this part of the backplate. Because of the assumed constant strain state in the element type used in the axisymmetric part, the stresses may be regarded as acting at the centroid of the element. The centroidal points of elements where stresses have been taken are shown by 'X' in Figure 6.1.

Both the circumferential and radial stresses at the rear of the backplate show good correlation with the results of Chan and Henrywood. As the axisymmetric assumption is not completely accurate and the assumed constant stress state in the axisymmetric element type gives rise to difficulties of positioning the calculated stresses, the accuracy of the stresses in this part is lower than that of the calculated stresses in the upper part of the backplate. The calculated stresses in the upper part lie very close to those obtained by Chan and Henrywood. They have not presented the stresses beneath blades at the front of the backplate. So the stresses, calculated here, have been compared with the finite difference results only. The results show good agreement except at the very lowest part of the back plate. This may be due to the elements in this part being influenced by their wide spread and the range of stresses encompassed by one element.

The blade stresses at the root show good correlation between the present analysis and the finite element analysis of Chan and Henrywood. The blade stresses at the leading
edge do not agree as well as those at the root. This is probably due to the influence of the 'mixed' shell elements which tend to produce spurious stresses at the stress free boundaries. However the general shape of the graph and the maximum stress obtained by the present method agree reasonably well with those of Chan and Henrywood's finite element analysis.

Thus the present analysis appears to be reasonably reliable because of good general agreement between the results obtained by this analysis and those by Chan and Henrywood's approach which is suitable for the analysis of radial rotors. The agreement with experimental results is reasonable considering the complexity of the application of the photoelastic method in this sort of problem.

Thus the above analysis establishes the reliability of the method used by the writer and leads to its application to more general rotors and other types of structure.
7. THE ANALYSIS OF GENERAL CURVED VANE IMPELLERS

The analysis of radial flow impellers presented in Chapter 6 and good agreement obtained there between the results of the writer and the results of other investigators gives one confidence in the reliability of the method proposed by the writer and its applicability to the analysis of general curved vane rotors. In this chapter the analyses of two such rotors are presented. As in the previous case, only the centrifugal loading will be considered since the aerodynamic loading is known to be comparatively small.

7.1 The Analysis of the Rotor Type BD1

This rotor, shown in Figure 7.1a, was supplied by Bryan Donkin & Co. Ltd., of Chesterfield. Because this rotor has been produced for commercial purposes, the exact details of the rotor cannot be given.

It was first analysed theoretically and then experiments with strain gauges were carried out for verification of the theoretical results.

7.1a Theoretical Analysis of the Rotor Type BD1

The rotor was divided into six parts as shown in Figure 7.1a. The parts (1), (2) and (3) constitute the back shroud; part (4) is the vane; and the parts (5) and (6) form the front shroud. The parts (1) and (6) were treated by axisymmetric solid elements (Chapter 11.4b); the parts (2), (3), (4) and (5) were treated by shell elements (Chapter 11.4c).
Identical Equally Spaced Vanes

'CCC' on vane is a free boundary

Figure 7.1a: Rotor Type BD1
It should be mentioned here that the line 'CCC' on vane (Figure 7.1a) is a free boundary, which justifies the use of axisymmetric elements in part (6).

As the element type used in the parts (2), (3), (4) and (5) is the same, and these parts have common nodes along boundaries, their individual stiffness matrices are combined automatically. However part (1) was connected to part (2) by a constraint matrix and similarly part (5) to part (6).

Because of symmetry only one of the identical segments, each containing a vane, was considered. The boundary conditions were imposed by using the constraint matrix method previously described.

The finite element meshes used in different parts of the structure are shown in Figure 7.1b to 7.1e (Chapter 11.6). According to the manufacturers' experience the most highly stressed parts are the front shroud and the blade. For this reason coarse meshes were used in parts (1), (2) and (3). It is difficult to approximate parts of variable thickness by large constant thickness elements. Because of this and also the existence of high stresses in the blade, a finer mesh was used in part (4). As the general stress level in the front shroud is comparatively high, finer meshes were also used in parts (5) and (6). The total effective number of degrees of freedom was 525.

The results obtained are presented in Graphs 7.1a to 7.1e. They show some expected patterns, for example, part (3) (the sealing ring) has some influence on the blade stresses in its vicinity and the influence of 'CCC' as a free boundary gives
rise to 'kinks' in the blade stresses (see Graph 7.3) at the point of attachment of the front shroud to the blade. The stresses in some localities exceed the yield stress in tension at the peak running speed. This is a very common experience with high speed rotating structures. The general stress level in the front shroud is higher than in other parts of the rotor.

The stress patterns generally agree with the manufacturers' experience. But the calculated stresses in the front shroud are higher than their expectations. The "equivalent Von Mises Stress" \(=\sqrt{\sigma_r^2 + \sigma_\theta^2 + (\sigma_r - \sigma_\theta)^2}/(\sqrt{2} \times \beta)\) in the inside of the front shroud was plotted in Graph 7.1a. The "equivalent Von Mises Yield Stress" \(= \sigma_c/\beta, \text{ where } \sigma_c = \text{Yield Stress in simple tension}\) attains the peak value in the graph at a speed of just over 10,000 rpm. From the above the yielding is likely to take place in the front shroud at a peripheral speed of about 1215 ft/second, which is higher than the peak running speed. The above yield criterion is likely to give a conservative estimate. Thus theoretically it appears that at the peak running speed there is a reasonable margin of safety.

It can be seen that the graphs of stresses on the front shroud show some "disturbances" near the junction between part (5) and part (6). This is the likely effect of the use of the constraint matrix to enforce compatibility of displacements at the common interface of part (5) and part (6).

The stresses in the back shroud were found to be generally lower than those in the front shroud, although high stresses were calculated at a few places. These may be due to structural
idealisation with very large elements.

The radial displacements along 'ZZ' of the bore is plotted in Graph 7.1f (Chapter 11.7) which shows conical deformation of the bore. This agrees with the manufacturer's experience.

7.1b Experimental Verification of the Theoretical Results of Rotor Type BD1

At the start of this project, it was expected that the manufacturer's test rig could be used for comprehensive strain gauge tests to verify the theoretical results. The experiments were to be in one or two stages depending upon the availability of time and facilities. In the first stage the strains at a few discrete points were to be "measured" with strain gauges. The purposes of this test were to be:

1. to check whether the calculated strains at these points compared favourably with the measured strains,
2. to check whether the axisymmetric assumption at the common boundary of the parts (5) and (6) was true (if the axisymmetric assumption at this point was untrue, then the strain gauge results were expected to give an indication of the extent of the portion behaving axisymmetrically),
3. to investigate the influence of constraint matrices on the stresses in the vicinity.

If the preliminary investigations gave encouraging results, then a more comprehensive set of strain gauge tests were to be carried out to check the theoretical stresses in various
parts of the structure.

Although strain gauges were fixed (positions shown in Figure 7.1b, 7.1d and 7.1e in Chapter 11.6) for the preliminary test, the facilities of the manufacturers were not available. The test had to be postponed indefinitely as the Department of Mechanical Engineering of this University could not allocate funds at that stage to build a test rig and buy other equipment. However, more recently, enough money was made available for building a temporary test rig and other necessary equipment, and the writer designed a rig which was quickly built by the technical staff of the Department. A slip ring unit with eight channels was made at the Department and a brush unit was bought to suit. A half bridge circuit was used for strain gauge connections. The following are the specifications of the equipment used:

1. the Test Rig: designed by the writer and built in the Department (shown in Figures 7.1f and 7.1g in Chapter 11.8),
2. the Strain Recorder: "Tequipment" Strain Recorder (Ref. No. B382270),
3. the Driving Motor: "Crofts" 5h.p., variable speed motor, speed ratio of 1940/388,
4. the Slip Ring Unit: designed by the writer and made in the Department,
5. The Brush Unit: 8-channel IDM brush unit,
6. Strain Gauges: etched foil strain gauges of 120Ω resistance, 6mm gauge length, and 2.13 gauge factor—temperature compensated for steel.
The inlet and outlet of the impeller were blocked as shown in Figure 7.1f (Chapter 11.8) in order to reduce wind resistance. It was possible to attain speeds of up to 1500rpm, beyond which brush noise made it too difficult to read the recorded strains. It was not suitable for long runs as the slip rings became hot and the perspex insulating rings started to deform.

Good signals and consistent strain readings were obtained up to 1500 rpm. These strain readings were plotted in the Graph 7.1g (Chapter 11.7) and the strains in various positions (shown in Figure 7.1b, 7.1d and 7.1e in Chapter 11.6) at a speed of 1000rpm have been tabulated in Table 7.1 along with the theoretically calculated strains.

Table 7.1: Comparison of Strain Gauge Results with Theoretical Results for the Rotor Type BD1.

<table>
<thead>
<tr>
<th>Gauge No.</th>
<th>Measured Strain at 1000 rpm (µin/in)</th>
<th>Calculated Strain at 1000 rpm (µin/in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>21</td>
<td>24</td>
</tr>
<tr>
<td>2</td>
<td>8</td>
<td>6</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>7</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>5^n</td>
</tr>
<tr>
<td>5</td>
<td>36</td>
<td>32^n</td>
</tr>
<tr>
<td>6</td>
<td>18</td>
<td>21^n</td>
</tr>
<tr>
<td>7</td>
<td>40</td>
<td>32^n</td>
</tr>
<tr>
<td>8</td>
<td>27</td>
<td>25^n</td>
</tr>
</tbody>
</table>

* from axisymmetric element  
$ from shell element  
n nearest point to the gauge where calculated stresses were available.
Table 7.1 shows reasonable agreement between theoretical and experimental results. Considering the difficulty of accurately positioning and aligning the strain gauges, the likely difference between actual dimensions and the dimensions on drawings, the probable experimental errors\textsuperscript{27,28} involved, the agreement is satisfactory. It should be noted that the lower level strains show greater discrepancies.

The readings of gauges 5 and 7 show some differences indicating that the axisymmetric assumption at the junction between the parts (5) and (6) is not quite correct. In spite of this, strains predicted by axisymmetric analysis of the part (6) compare reasonably satisfactorily with actual strains. The strains indicated by the part (5) (treated as shell) at gauge positions 4, 5, 6 and 8 show fair agreement with experimental strains in spite of the use of the constraint matrix to enforce compatibility between nodal displacements of the parts (5) and (6) at the common junction.

Thus on the whole, the analysis of this rotor by the present method was considered to be successful.

7.2 The Analysis of the Rotor Type JH1

With the successful analysis of the rotor type BDI described above, the writer looked for practical rotors of complexity for analysis. The rotor type, shown in Figure 7.2a, was chosen by the present investigator for analysis because of its complexity and availability of experimental results from the makers. This rotor was supplied by James...
Howden and Company Limited of Glasgow, who produce it commercially to suit customer requirements. Because of its present commercial use, more exact details of the rotor than those given in Figures 7.2a and 7.2b cannot be given.

The rotor has "thin" shrouds and blade skin compared to its size. There are two stiffeners (see blade details in Figure 7.2b), inside the blade profile, going across the span. The stiffeners themselves are "thin" plates welded to the trailing blade skin for full length but only a limited length at each end of the leading blade skin. The conical supporting disc for the front shroud is also thin. Thus, except for the hub, the whole structure could be described by shell elements. The hub is far away from the vanes, so that the effect of the vanes in producing non-axisymmetric stress pattern is unlikely to be felt in this part. Consequently it can be fairly well represented by axisymmetric solid elements.

Initially, for the whole structure, the writer produced a finite element mesh which he considered to be of a reasonable degree of refinement for a satisfactory solution. But the estimated computer time could not be available in the period of time available to the writer for the analysis of this structure, because of heavy demand for time in the University Computer. So the writer then produced a much coarser mesh for the structure. In particular, very large elements were used in parts (2) and (3), as the interest mainly lay in the stress distribution in the shrouds near the blades, and the blades themselves. Again, because of symmetry, only one of
Leading Blade Face  Trailing Blade Face  Back Shroud

12 equally spaced identical vanes

N.B. The 'Longitudinal' and 'Normal' stresses in the blade denote the stresses in the directions A-A', B-B', C-C' & normal to those directions.

Figure 7.2a: The Rotor Type JH1
Figure 7.2b: The Blade Profile in Rotor Type JH1
the twelve identical segments, each with a blade, was analysed. The finite element meshes in different parts of the structure are shown in Figures 7.2c to 7.2f (Chapter 11.6). The mesh in part (6), the front shroud, was almost identical to that in part (4), the back shroud. It will be noticed in Figure 7.2e that on the shrouds, the line between blades has been kept away from the boundary in order to reduce the effect of the equations of constraint imposing the boundary conditions, described below, in the stress distribution in the vicinity of the line.

The stiffness matrices of the parts (2), (3), (4), (5) and (6) were combined together automatically because of the same element type being used in each part and common nodal points at mutual boundaries. However, because of incompatible element types, the stiffness matrix of part (1) was combined with those of part (2) and part (3) by the use of two constraint matrices.

As in the previous cases, the boundary conditions were imposed by equating the nodal displacements of one edge of each shroud to the corresponding nodal displacements of the other edge.

The calculated results were plotted in Graphs 7.2a to 7.2s (Chapter 11.7). The makers had done some experimental investigation of the stress distribution in the front shroud and the blades using 45° strain gauge rosettes. These experimental stresses were plotted in the Graphs 7.2a to 7.2p alongside the theoretical stresses. The Graphs 7.2a, 7.2b and 7.2c present the circumferential stresses in the front shroud:
one inch in front of the blade, between blades, and one inch behind the blade. Considering the finite element mesh used in the shrouds, the correlation between theoretical and experimental results is acceptable.

The graphs 7.2d to 7.2p present the normal and longitudinal theoretical stresses in the blade at half an inch from the shrouds, and between shrouds. The experimental stresses on the outside surface along these lines are also presented. Unfortunately, the experimental points are far apart and give no indication of the influence of the stiffeners on the skin stresses. The writer feels that the centrifugal forces due to the stiffeners would affect the skin stresses in their vicinity. However, as the experimental points are placed between vanes, it cannot be verified by the experimental results. The theoretical results show fluctuations of stresses which seem to be due to the presence of stiffeners. The fluctuations are more between the shrouds than near the shrouds. The experimental stresses between shrouds (line BB in Figure 7.2a) on both the leading and trailing blade surfaces appear to be approximately the average of the theoretical stresses on the outside and the inside skin indicating the possibility of an induced spurious bending stress in the theoretical calculations. The writer thinks that the following is a possible explanation for this. It will be remembered that the stiffeners are welded to the leading blade skin only a limited length at each end. In the idealised structure for theoretical analysis the stiffeners have no connection with the blade leading skin for a large part and
consequently it allows the blade skins to sag under the centrifugal loading producing a complex distribution of curvatures giving rise to bending stresses. In practice, however, the stiffeners are likely to support the leading blade skin preventing both skins from sagging. It can be seen that the stress fluctuations at the trailing skin are comparatively less, lending support to the above view.

Near the shrouds, the experimental and theoretical stresses on the outside of both the leading and trailing blade surfaces seem to have the same pattern, although quantitatively the agreement is not good and the theoretical stresses continue to fluctuate. Of course, after such wide variations in the middle of the blade, the blade stresses could not be expected to agree well with the experimental results near the shrouds. The stress distributions in the blade only half an inch from the shrouds are likely to be very complex and a more refined mesh would be necessary for satisfactory results.

The stress distributions in the back shroud (Graphs 7.2q to 7.2s) have the same pattern as in the front shroud giving reasonable credibility in them.

The agreement between experimental and theoretical results in the front shroud with only a coarse mesh shows that the method is applicable to this problem. Considering the complexity of the problem and the degree of mesh refinement, the fact that the experimental and theoretical blade stresses were of the same order is encouraging. A further investigation of the blade stresses should be carried out.
with a finer mesh and, possibly, with the use of equations of constraint to simulate the supports given by the stiffeners to the blade skins. Moreover the writer feels that some more experimental work ought to be carried out for a more detailed picture of the stress distribution in the blade. If possible, more suitable shell elements ought to be used in this type of structure, especially in blades, as the element used here is not well suited for closed ended box type structures$^{15}$. 

On the whole the method seems a promising one for this sort of structure as the problem was analysed with 1493 effective degrees of freedom, and with the introduction of sufficient extra degrees of freedom it will remain an economic proposition.
Although the purpose of this project was to develop a technique generally applicable to rotors of the type shown in Figure 1.1, and this rotor and the other, with curved vanes and shrouds analysed by the writer were reasonably adequately represented by axisymmetric solid elements and flat plate shell elements, it may be necessary to use fully three-dimensional elements for proper representation of other types of rotors. As there was no such element available in the Loughborough University Finite Element System, it was decided to develop one.

In the context of analysis of rotors, a cylindrical solid element described by cylindrical coordinates was thought to be most suitable considering the ease of finite element mesh generation and data preparation. Also it was felt that the element should be simple and comparatively inexpensive in terms of computation time. As triangular elements are best suited for representing structures with complicated geometry, this type of shape was chosen for the present element. Because much of the algebra can be performed by hand, the right-angled triangular elements require less computer time, and thus for reasons of economy the element formulation developed here is of this type.

A right-angled triangle can adopt one of the four attitudes shown in Figure 8.1.

The stiffness matrices of the elements (b) and (d) can be obtained from those of the elements (a) and (c)
respectively by means of simple transformation matrices and so it is only necessary to develop the stiffness matrices of the elements (a) and (c). Here the development of the element stiffness matrix is shown considering the element (a). Figure 8.2 shows one such element.

\[
\begin{align*}
\sigma_z &= \beta_1 + \beta_2 r + \beta_3 \theta + \beta_4 \theta^2 + \beta_5 z \\
\sigma_\theta &= \beta_{11} + \beta_{12} r + \beta_{13} \theta + \beta_{14} \theta^2 + \beta_{15} z \\
\sigma_r &= \beta_{11} + \frac{\beta_{12} r}{2} + \beta_{13} \theta + \beta_{14} (1+\theta^2) + \beta_{15} z \\
\tau_{r\theta} &= -\frac{\beta_{13} r}{2} - \beta_{14} \theta - \frac{\beta_{15} r}{3} \\
\tau_{z\theta} &= \beta_6 + \beta_7 r + \beta_8 \theta + \beta_9 \theta^2 + \beta_{10} z \\
\tau_{zr} &= -\beta_8 - 2\beta_9 \theta - \frac{\beta_5 z}{2} r
\end{align*}
\]

As the shell elements in the system are developed by the mixed method and also the mixed method makes the problem of inter-element displacement continuity easier to deal with, this method (see Chapter 11.1b) has been chosen here for the development of the stiffness matrix of this element.

Referring to Chapter 11.1b the following stress fields were chosen:
Inside diameter: = 1"
Outside diameter: = 4"
Thickness of disc = 0.5"

Loading:
1. A uniformly distributed load of 18,000 lbf acting radially outwards along the circumference.
2. A uniformly distributed load of 9600 lbf acting vertically downwards along the circumference.
These satisfy the following equilibrium equations for a three-dimensional solid in cylindrical coordinates:

\[
\begin{align*}
\frac{1}{r} \frac{\partial}{\partial r} (r \sigma_r) + \frac{1}{r} \frac{\partial}{\partial \theta} (r \tau_{r\theta}) + \frac{\partial}{\partial z} (\tau_{rz}) - \frac{\sigma_0}{r} &= 0 \\
\frac{1}{r^2} \frac{\partial}{\partial r} (r^2 \tau_{r\theta}) + \frac{1}{r} \frac{\partial}{\partial \theta} (\tau_{z\theta}) &= 0 \\
\frac{1}{r} \left( \frac{\partial}{\partial r} (r \tau_{zr}) \right) + \frac{1}{r} \frac{\partial}{\partial \theta} (\tau_{z\theta}) + \frac{\partial}{\partial z} (\sigma_z) &= 0
\end{align*}
\] ....(8.2)

So the P-matrix is given by

\[
[P] = \begin{bmatrix}
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & r & \theta (1+\theta^2) \\
1 & r & \theta & \theta^2 & z & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & -r/2 & 0 & -1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 1 & r & \theta & \theta^2 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\] (8.3)

The stress-strain relationship is given by the following equations:

\[
\begin{bmatrix}
\epsilon_r \\
\epsilon_\theta \\
\epsilon_z \\
\gamma_{r\theta} \\
\gamma_{rz} \\
\gamma_{\theta z}
\end{bmatrix} = \frac{1}{E} \begin{bmatrix}
1 & -\nu & -\nu & 0 & 0 & 0 \\
-\nu & 1 & -\nu & 0 & 0 & 0 \\
-\nu & -\nu & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\
0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\
0 & 0 & 0 & 0 & 0 & 2(1+\nu)
\end{bmatrix} \begin{bmatrix}
\sigma_r \\
\sigma_\theta \\
\sigma_z \\
\tau_{r\theta} \\
\tau_{rz} \\
\tau_{\theta z}
\end{bmatrix}
\] (8.4)

or \( \{\epsilon\} = [N]\{\sigma\} \)

Then the H-matrix is obtained from the equation

\[
\{\sigma\} = [H]\{\epsilon\}
\]
\[ [H] = \int_V [P]^t [N] [P] dV \] .................................. (8.5)

The surface traction components are obtained by substituting the equations of surfaces in the P-matrix in the following equations:

\[
\begin{bmatrix}
\frac{n}{T_r} \\
\frac{n}{T_\theta} \\
\frac{n}{T_z}
\end{bmatrix} = \begin{bmatrix}
n_1 & 0 & 0 & n_2 & n_3 & 0 \\
0 & n_2 & 0 & n_1 & 0 & n_3 \\
0 & 0 & n_3 & 0 & n_1 & n_2
\end{bmatrix} \begin{bmatrix}
\{\beta\}
\end{bmatrix} = \begin{bmatrix}
[R_1] \{\beta\}
\end{bmatrix}
\]

(8.6)

as \( \{\sigma\} = [P] \{\beta\} \)

\( n_1, n_2, n_3 \) are direction cosines of the surface normal \( n \).

Thus the part of the R-matrix for a particular face is obtained.

The interpolation matrix, \( L \), is difficult to obtain explicitly in this case. So for a particular surface a displacement function is chosen such that its displacement components in the direction of the surface traction components are given by

\[
\begin{bmatrix}
u \\
v \\
w
\end{bmatrix} = [M_1] \{a\} \] ................................ (8.7)

where \( a \)'s are unknown coefficients.

Now if \( \{q_1\} \) is the nodal displacement vector of the surface then
\{a\} = [M_2]^{-1}\{q_1\} \hspace{1cm} (8.8)

where $[M_2]$ contains coordinates of the surface.

Then

\[
\begin{bmatrix}
  u \\
  v \\
  w
\end{bmatrix} = [M_1][M_2]^{-1}\{q_1\} = [L_1]\{q_1\} \hspace{1cm} (8.9)
\]

As an example, for face(1) (see Figure 8.2)

\[
\begin{bmatrix}
  u \\
  v \\
  w
\end{bmatrix} = \begin{bmatrix}
  1 & r & z & 0 & 0 & 0 & 0 & 0 & 0 \\
  0 & 0 & 0 & 1 & r & z & 0 & 0 & 0 \\
  0 & 0 & 0 & 0 & 0 & 0 & 1 & r & z
\end{bmatrix} \begin{bmatrix}
  a_1 \\
  a_2 \\
  a_3 \\
  a_4 \\
  a_5 \\
  a_6
\end{bmatrix} \hspace{1cm} (8.10)
\]

\[
= [M_1]\{a\}
\]

and

\[
\{a\} = \begin{bmatrix}
  1 & r_1 & z_1 & 0 & 0 & 0 & 0 & 0 & 0 \\
  0 & 0 & 0 & 1 & r_1 & z_1 & 0 & 0 & 0 \\
  0 & 0 & 0 & 0 & 0 & 0 & 1 & r_1 & z_1 \\
  0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
  1 & r_2 & z_2 & 0 & 0 & 0 & 0 & 0 & 0 \\
  0 & 0 & 0 & 1 & r_2 & z_2 & 0 & 0 & 0 \\
  0 & 0 & 0 & 0 & 0 & 0 & 1 & r_2 & z_2 \\
  1 & r_3 & z_3 & 0 & 0 & 0 & 0 & 0 & 0 \\
  0 & 0 & 0 & 1 & r_3 & z_3 & 0 & 0 & 0 \\
  0 & 0 & 0 & 0 & 0 & 0 & 1 & r_3 & z_3
\end{bmatrix}^{-1} \begin{bmatrix}
  u_1 \\
  v_1 \\
  w_1 \\
  u_2 \\
  v_2 \\
  w_2 \\
  u_3 \\
  v_3 \\
  w_3
\end{bmatrix} \hspace{1cm} (8.11)
\]
Now from equations (8.6) and (8.9) the part of the T-matrix concerning a particular surface under consideration can be obtained from

\[ [T_i] = \int_S [R_i]^T[L_i] dS \] ........................(8.12)

superimposing the contributions from all surfaces the T-matrix is completed.

Then the element stiffness matrix is obtained from

\[ [K_e] = [T]^T[H]^{-1}[T] \] ........................(8.13)

It should be noted here that the above approach could be used for developing triangular elements of any shape, but the integrations might have to be performed numerically.

The above formulation was translated into a computer program and the convergence of results obtained from this element was studied by investigating the system shown in Figure 8.3. The system was analysed first by the three-dimensional cylindrical solid element with a type of mesh shown in Figure 8.4a (Chapter 11.6), and then it was analysed with axisymmetric elements (Chapter 11.4b) with a type of mesh shown in Figure 8.4b (Chapter 11.6). The results obtained in both cases are compared with each other by plotting the vertical and radial deflections of a point A on the periphery in Graphs 8.1 and 8.2 (see Chapter 11.7). As the axisymmetric element is well tested and widely used, it can be safely used as a criterion.
Graphs 8.1 and 8.2 show good comparison between the results obtained in the two cases. Although the results obtained by the three-dimensional cylindrical solid element are poorer than those obtained by the axisymmetric element for a given mesh, the rate of convergence of the three-dimensional element is acceptable.

Although more tests should be done in order to establish the expected accuracy of results obtained by the present element in various types of problem, and in particular its use for non-axisymmetric loading should be investigated, the reasonable correlation between the results obtained by this element and those by the axisymmetric element, which is the nearest equivalent in the displacement approach for axisymmetric analysis, gives some degree of confidence in it. It should be noted here that in a full ring the displacement on the common interface at the beginning and the end of the ring may be found to be incompatible. In such cases, equations of constraint may be used to enforce compatibility.
Use of Constraint Matrices in the Development of Finite Element Stiffness Matrix

Constraint matrices are frequently used in the development of finite element stiffness matrices in various forms. For example, transformation matrices, which can be described as a particular type of the constraint matrix where the number of generalised degrees of freedom after constraining remains the same as before constraining, are used many times to facilitate computation by first developing an element stiffness matrix in terms of a suitable 'local' coordinate system and then expressing the 'local' stiffness matrix in terms of the 'global' coordinates. In other cases the constraint matrices are used to reduce the number of generalised degrees of freedom of an element. For example, in order to achieve inter-element displacement compatibility in a triangular plate bending formulation it may be necessary to introduce an additional node in the middle of each side of the triangle with just one rotational degree of freedom. But certain programming difficulties are encountered with elements having different degrees of freedom at different nodes. So it may be desirable to eliminate the mid-side nodes after the element stiffness matrix has been formulated. This can be done by establishing equations of constraint relating the generalised degree of freedom of the mid-side nodes to those of the corner nodes. From these equations constraint matrices can be developed and used to reduce the stiffness matrix so that the element is described by the
degrees of freedom of the corner nodes only.

The writer here proposes another use of a transformation matrix in the formulation of an element stiffness matrix, for a slightly different purpose. As has been mentioned above, sometimes it is necessary to use element formulations using different nodes with different numbers of degrees of freedom, in order to maintain inter-element displacement continuity, e.g. in the triangular plate bending formulation. In general this is done by assigning, at the start, different degrees of freedom to different nodes. The writer feels that this should be avoided and, at the stage of an element stiffness matrix formulation, all nodes should have the same degrees of freedom. After the element stiffness matrix has been formulated, it may be transferred to any feasible set of generalised degrees of freedom distributed in a suitable manner to the nodes.

This is illustrated by considering the formulation of a triangular plate bending element by the displacement method with different nodes having different numbers of degrees of freedom.

The following displacement field is chosen for $w$:

$$w = a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 x y + a_6 y^2 + a_7 x^3$$

$$+ a_8 x^2 y + a_9 x y^2 + a_{10} y^3 + a_{11} x y^3 + a_{12} x y^3$$

... (9.1)

Now

$$\theta_x = -\frac{\partial w}{\partial y} \quad \text{and} \quad \theta_y = \frac{\partial w}{\partial x}$$
The nodes of the element are initially assumed to be 1, 2, 3 and 4 (see Figure 9.1), each with three degrees of freedom \((w, \theta_x, \theta_y)\). 4 is any convenient point in the element and 1, 2 and 3 are nodes on sides depending on the position of 4.

\(a\)'s can be described in terms of the nodal displacement vector,

\[
\{\delta_e\} = \{w_1 \theta_{x_1} \theta_{y_1} w_2 \theta_{x_2} \theta_{y_2} w_3 \theta_{x_3} \theta_{y_3} w_4 \theta_{x_4} \theta_{y_4}\}^T,
\]

such that

\[
\{a\} = [B]^{-1} \{\delta_e\}
\]

where \([B]\) contains the coordinates of nodes. As the node 4 is any point in the element, \([B]\) will not in general be singular.

Thus

\[
\begin{bmatrix}
w \\
\theta_x \\
\theta_y
\end{bmatrix} = [A] [B]^{-1} \{\delta_e\}
\]
Now the strain energy per unit volume is given by

\[
U_0 = \frac{1}{2} \left[ M_x, M_y, M_{xy} \right] \begin{pmatrix}
\frac{\partial^2 W}{\partial x^2} \\
\frac{\partial^2 W}{\partial y^2} \\
\frac{\partial^2 W}{\partial x \partial y}
\end{pmatrix}
\]

\[
= \frac{1}{2} \begin{pmatrix}
\frac{\partial^2 W}{\partial x^2} \\
\frac{\partial^2 W}{\partial y^2} \\
\frac{\partial^2 W}{\partial x \partial y}
\end{pmatrix}^T \times \frac{E t^3}{12(1-\nu^2)} \times \begin{pmatrix}
1 & \nu & 0 \\
\nu & 1 & 0 \\
0 & 0 & \frac{(1-\nu)}{2}
\end{pmatrix} \begin{pmatrix}
\frac{\partial^2 W}{\partial x^2} \\
\frac{\partial^2 W}{\partial y^2} \\
\frac{\partial^2 W}{\partial x \partial y}
\end{pmatrix}
\]

\[
= \frac{1}{2} \{\varepsilon\}^T [N] \{\varepsilon\}
\]

Now \(\{\varepsilon\} = [Q] \{\alpha\}\) ..........................(9.4)

where

\[
[Q] = \begin{pmatrix}
0 & 0 & 0 & -2 & 0 & 0 & -6 & -2y & 0 & 0 & -6xy & 0 \\
0 & 0 & 0 & 0 & 0 & -2 & 0 & 0 & -2x & 6y & 0 & -6xy \\
0 & 0 & 0 & 0 & 2 & 0 & 0 & 4x & 4y & 0 & 6x^2 & 6y^2
\end{pmatrix}
\]

Now

\[
\{\varepsilon\} = [Q] \{\alpha\} = [Q] [B]^{-1} \{\delta_e\}
\]

so that

\[
U_0 = \frac{1}{2} \{\delta_e\}^T \left([Q] [B]^{-1}\right)^T [N] \left([Q] [B]^{-1}\right) \{\delta_e\}
\]

and
\[ \frac{\partial}{\partial \delta_e} \iint U_0 \, dx \, dy = \{ [B]^{-1} \}^T \left( \iint [Q]^T [N] [Q] \, dx \, dy \right) [B]^{-1} \{ \delta_e \} \quad (9.8) \]

Thus the element stiffness matrix, \([K_e]\), is obtained in terms of the displacement vector,

\[ \{ w_1 \, \theta x_1 \, \theta y_1 \, w_2 \, \theta x_2 \, \theta y_2 \, w_3 \, \theta x_3 \, \theta y_3 \, w_4 \, \theta x_4 \, \theta y_4 \}^T \].

Now in order to achieve full continuity of displacements along common inter-element boundaries, it is necessary to transform \([K_e]\) in terms of the nodal displacement vector

\[ \{ w_a \, \theta x_a \, \theta y_a \, w_b \, \theta x_b \, \theta y_b \, w_c \, \theta x_c \, \theta y_c \, w_d \}^T \]

(see Figure 9.2).

Assuming a cubic variation of \( w \), the vertical displacement, along the direction \( s \), it can be given by

\[ w = a_1 + a_2 s + a_3 s^2 + a_4 s^3 \quad \ldots \ldots \ldots (9.9) \]

and

\[ \theta_s = \frac{\partial w}{\partial s} = a_2 + 2a_3 s + 3a_4 s^2 \quad \ldots \ldots \ldots (9.10) \]

Substituting the boundary conditions in terms of \( w_a, \theta s_a, w_b \) and \( \theta s_b \), \( a \)'s can be determined. Thus

\[ w = H_1 w_a + H_2 w_b + H_3 \theta s_a + H_4 \theta s_b \quad \ldots \ldots \ldots (9.11) \]

and

\[ \theta s' = I_1 w_a + I_2 w_b + I_3 \theta s_a + I_4 \theta s_b \quad \ldots \ldots \ldots (9.12) \]

where
\[
\begin{align*}
H_1 &= 1 - \frac{3s^2}{\kappa^2} + \frac{2s^3}{\kappa^3} \\
H_2 &= \frac{3s^2}{\kappa^2} - \frac{2s^3}{\kappa^3} \\
H_3 &= s \times \left( -\frac{s}{\kappa} + \frac{s^2}{\kappa^2} \right) \\
H_4 &= s \times \left( -\frac{s}{\kappa} + \frac{s^2}{\kappa^2} \right)
\end{align*}
\]

and
\[
\begin{align*}
I_1 &= \left( \frac{s}{\kappa} - 1 \right) \left( \frac{6s}{\kappa^2} \right) \\
I_2 &= \left( 1 - \frac{s}{\kappa} \right) \left( \frac{6s}{\kappa^2} \right) \\
I_3 &= 1 - \frac{4s}{\kappa} + \frac{3s^2}{\kappa^2} \\
I_4 &= \frac{3s^2}{\kappa^2} - \frac{2s}{\kappa}
\end{align*}
\]

Now considering Figure 9.4

\[
\begin{align*}
\theta_{xi} &= \theta_{si} \cos \theta - \theta_{si} \sin \theta \\
\theta_{yi} &= \theta_{si} \sin \theta + \theta_{si} \cos \theta
\end{align*}
\]

Thus from equations (9.11), (9.12) and (9.15), the displacement vector, \( \{ \delta_f \} \) (= \( \{ w_1 \ \theta_{x1} \ \theta_{y1} \ w_2 \ \theta_{x2} \ \theta_{y2} \ w_3 \ \theta_{x3} \ \theta_{y3} \}^T \)), may be given by

\[
\{ \delta_f \} = [T_1] [T_2] \{ \delta_g \}. \tag{9.16}
\]

where \( \{ \delta_g \} \) = \( \{ w_a \ \theta_{xa} \ \theta_{ya} \ \theta_{si} \ w_b \ \theta_{xb} \ \theta_{yb} \ \theta_{si} \ w_c \ \theta_{xc} \ \theta_{si} \}^T \). \tag{9.17}

and \([T_2]\) is obtained by using equations (9.11) and (9.12)
Figure 9.3

Figure 9.4

Figure 9.5
while $[T_e]$ is obtained from equation (9.15).

Now $w_4, \theta_{x4}, \theta_{y4}$ can be expressed in terms of $\theta_{s1}$,
$\theta_{s1}, \theta_{s2}, \theta_{s2}, \theta_{s3}, \theta_{s3}$ such that
\[
\begin{bmatrix}
w_4 \\
\theta_{x4} \\
\theta_{y4}
\end{bmatrix} = [T_3] \{g\} 
\] ............ (9.18)

Then the original displacement vector,
$\{\delta_e\} = \{w_1 \theta_{x1} \theta_{y1} w_2 \theta_{x2} \theta_{y2} w_3 \theta_{x3} \theta_{y3} w_4 \theta_{x4} \theta_{y4}\}^T$

can be given in terms of the new displacement vector,
$\{\delta_g\}$ (equation 9.17), such that
\[
\{\delta_e\} = [T_1] [T_2] [T_3] \{\delta_g\} 
\] = $[C] \{\delta_g\}$

............ (9.19)

The element stiffness matrix, $[K_g]$, in terms of the new displacement vector, $\{\delta_g\}$, is obtained from
\[
[K_g] = [C]^T [K_e] [C] 
\] ............ (9.20)

where $[K_e]$ is the element stiffness matrix in terms of the displacement vector $\{\delta_e\}$.

A computer program was written to generate the above plate bending stiffness matrix and a simply supported square plate (shown in Figure 9.5) under a central concentrated load was analysed by this element with a mesh shown in Figure 9.6 (Chapter 11.6).
The vertical central deflection, \( w \), calculated with the present element is compared in Table 9.1 with that obtained by the direct analytical method\(^2\).

<table>
<thead>
<tr>
<th></th>
<th>Direct Method</th>
<th>Present Finite Element Method</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>0.01160</td>
<td>0.00989</td>
</tr>
</tbody>
</table>

Table 9.1

The vertical central deflection \( = \alpha \frac{Pa^2}{D} \)

where \( \alpha = 2 \times \ell \)

\( P = \) the central load

\( D = \frac{Et^3}{12(1-v^2)} \) where \( t = \) plate thickness.

Although it is not possible to assess the general accuracy of the method from one set of results, it shows the validity of the method. The extra computational effort required for this element over similar plate bending elements is probably not justified but this presents an approach which may be found to be useful in some cases.
10. DISCUSSION AND CONCLUSIONS

The engineer's job is a difficult one requiring continuous search for innovation. The solution of most engineering problems is a gradual step by step procedure as there is rarely a complete solution of a real life problem. Most up to date techniques of yesterday become obsolete today and the engineer's answer to a problem is often an improvement over the previous one but not a complete solution. The writer has introduced a technique which has led him to such a step forward in the theoretical static analysis of complex centrifugal structures.

As has been mentioned in the introduction, no publications were to be found dealing with the theoretical analysis of complex rotors when the writer started his work on this project, and the purpose of this investigation was to develop a generally applicable economic method of analysing complex rotating structures. The finite element method could certainly be used in this problem by treating the whole structure with three-dimensional solid elements, but the computation time for such analysis would be excessive due to the necessity for placing several elements through the thickness of the thin sections and the consequent small size of the elements in the plane of the surface, bearing in mind that the rounding-off error becomes significant for elements with one dimension very much larger than the others, (as has been stated by Chan and Henrywood more recently). However in December 1971 Chan and Henrywood published their work on the
analysis of radial flow rotors, which was an improvement over the existing techniques. One of the typical problems analysed by them was also analysed by the writer (Chapter 6). The results obtained by the writer were in good agreement with their results. This demonstrates the reliability of the method used by the writer. Although they have not stated the number of unknowns they used for their analysis of this rotor, they have mentioned 785 and 1202 unknowns being used for their test problems. The writer used only 90 unknowns for his analysis. Thus assuming that they used a similar number of unknowns for the analysis of the rotor as for their test problems, the method used by the writer is certainly an improvement in saving the cost of analysis which was one of the main points of concern of the writer.

The writer does not claim that the element types used here are the most suitable. These were the elements available to him. The axisymmetric element type used here has been used for solving a wide range of similar problems and is quite dependable if used with a proper feeling for its limitations. However this is a simple constant strain element and therefore high stress gradients cannot be well represented by its use.

The shell element used here appears to be dependable although it has its limitations because the "average rotation" \( \frac{1}{2}(\frac{\partial u}{\partial x} - \frac{\partial u}{\partial y}) \) as a degree of freedom per node, implies zero shear strain. Moreover, it tends to produce spurious stresses at stress free boundaries. However it has been used in Reference (15) for the analysis of shell
structures, including some box type structures. It has been reported in the above reference that this element yielded good results for open ended box type structures, but not such good results for closed ended ones. From this it can be concluded that this element was suitable for the rotor type BD1, but not quite appropriate for the blades of the rotor type JH1. As a consequence some miscalculations are likely to occur in the stress distribution at the ends of the blades adjoining the shrouds. The blades and the back shroud of the rotor type BD1 are of variable thickness. Representing variable thickness shells by constant thickness elements somewhat misrepresents the problem. Moreover, as at the junction of two elements of different thicknesses, the stresses are discontinuous, the stress interpretation becomes difficult. In this thesis, simple averages of the stresses given by different elements at a common point were taken but this is not an entirely satisfactory procedure. The variable thickness shell elements would be most appropriate for proper analysis of such parts. Although there were no very highly curved parts in the rotors analysed by the writer, there are rotor structures with such parts requiring curved shell elements for satisfactory solution. The writer feels that some small parts in the blade of the rotor type BD1 should have been formulated as thick shells. Similar instances can be found in many practical rotors. Moreover, the junctions of shells is a problem by itself and there are elements\textsuperscript{15}, usually expensive in terms of computation time, which can represent the conditions at junctions better than
others. Such elements can be used in limited areas near junctions for better results without increasing the computation time significantly.

However, in spite of the absence of the element types mentioned above, required for truly satisfactory analysis, good agreement between experimental and theoretical results were obtained in the rotor type BD1 and in the shrouds of the rotor type JH1. The discrepancies between experimental and theoretical blade stresses in the rotor type JH1 could be accounted for by the properties of the shell elements used and errors in structural idealisation as described in Chapter 7.2.

The results of the above investigations have been quite informative. For example, the manufacturers of the rotor type BD1 had no clear picture of the effects of the sealing ring (Figure 7.1a) on the blade stresses in its vicinity and the influence of the free edge of the blade ('CCC' in Figure 7.1a) on the blade stresses, particularly near the point of attachment of the front shroud with the blade. These have been found to be significant, and should be considered at the design stage.

The analyses of the above rotor types were accomplished with only 525 and 1493 unknowns respectively. It should be mentioned here that the time taken by the operations involving constraint matrices is usually a small part of the total computation time. Considering the complexity of the problems involved, these figures are quite satisfactory and are well within the limits imposed by economic considerations. This
is an achievement which has been made possible by the use of
the constraint matrix technique enabling the most economic
of the suitable element types to be used in all parts of the
structure to be analysed. Although the writer did not have
many element types at his disposal for a comparative analysis,
the constraint matrix technique will allow the most appropriate
element type to be used for representing a particular segment
of a structure not only considering economy but also the
desired accuracy.

Thus it can be safely concluded that the constraint
matrix method used by the writer together with the use of a
variety of element types depending on their ability to
represent various conditions mentioned above is a technique
well suited for the analysis of complex rotors.

As three-dimensional elements will be required for
proper formulation of some segments of many structures, it
is essential to have a three-dimensional solid element in a
comprehensive system. A most suitable three-dimensional
element type for use in the analysis of rotors is one
developed in terms of a cylindrical coordinate system,
because of the ease of mesh generation and data preparation.
The element developed by the writer (Chapter 8) is of such
a type and has given good displacement convergence on axi-
symmetric problems. Because it was not planned at the initial
stages of this work, the investigation of complex three-
dimensional problems with this element could not be carried
out because of lack of time, but the writer feels confident
that it will give results of sufficient accuracy for most
problems.

Another use of the constraint matrix has been demonstrated in Chapter 9 in achieving inter-element displacement continuity in a plate bending "displacement" model with nodes on sides having only one degree of freedom but allowing all the nodes of an element to have the same degrees of freedom at the stage of the stiffness matrix formulation. This was not planned to be a part of the investigation and was pursued simply to illustrate the diversity of possible applications of the constraint matrix technique. As a result sufficient attention could not be concentrated on it to establish the usefulness of the element. However, the writer feels that the idea has been well illustrated and, to some extent, substantiated by the one result presented.

The constraint matrix technique has also been shown to be effective in enforcing inter-element compatibility of displacements in other situations (Chapter 5). As an extension of the case considered in Chapter 5, the constraint matrix can be used to represent areas of high stress gradient with sophisticated element types while using simpler element types elsewhere by enforcing displacement compatibility at the lines of transition from one element type to another.

Thus in the course of this investigation a general economic method has been developed for the analysis of complex rotors, a comprehensive set of element types has been accumulated for the above purpose with the introduction of the three-dimensional cylindrical solid element, and the constraint matrix technique has been demonstrated to be a
powerful tool with diverse applications in the field of the finite element analysis of structures.
11. **APPENDICES**

11.1 **The Finite Element Method**

A continuum contains an infinite number of points, each with three degrees of freedom, so that effectively it may be said to have an infinity of degrees of freedom. Formulations (of the governing equations for stress and displacement) in continuum mechanics must therefore be by means of partial differential equations. Real engineering problems rarely involve boundaries which are easy to express analytically and it is often impossible to arrive at an analytical solution to anything even closely approximating the real problem.

The Finite Element Method\(^7\) reduces problems with an infinite number of degrees of freedom to ones with a finite number of degrees of freedom, and thus makes many continuum mechanics problems easier to deal with.

For example, if the displacement properties of the tri-

![Figure 11.1:](image)

\(^7\) The Finite Element Method.
angular lamina shown in Figure 6.1 are expressed in terms of displacements at the three nodes, then its infinite number of degrees of freedom is replaced by only six degrees of freedom.

It is a versatile method and can be used to calculate stresses and deflections in two- or three-dimensional structures. Most of the original work on the Finite Element Method was done in the aircraft industry to solve complex problems of aircraft structures. Availability of large digital computers in the aircraft industry helped in the development of the Finite Element Method in the early stages. Now, however, work on the Finite Element Method is being carried out in many universities and other industries as well as aircraft industry itself. Although most of the basic problems regarding the Finite Element Method have been solved and a sound mathematical basis has been established, many new developments are taking place, particularly in its applications to various problems. The Finite Element Method is being increasingly used in the fields of fluid mechanics, visco-elasticity, plasticity, etc. as well as in elastic structural problems. Today there exist many large finite element system programs capable of handling a large range of problems.

The basic concept of the Finite Element Method is that a structure or continuum may be considered to be an assemblage of a finite number of individual structural components or elements. The individual components are connected at a number of points or nodes. For example, the plate shown
in Figure 11.2 can be divided into four triangular elements, connected together by nodes 1, 2, 3, 4 and 5. After dividing a structure into elements the physical properties of individual elements are determined. In the case of static elastic analysis, the physical property involved is stiffness or flexibility. As the elements are connected at nodal points, the elastic properties of the elements are described by the relationship of the applied forces at the nodes and the deflections due to these forces.

When the elastic properties of elements have been evaluated the problem is made to satisfy the following requirements:

(a) **Equilibrium:** the external forces acting on each node of each element must be in equilibrium with the internal forces of each element.

(b) **Compatibility:** the deformations of individual elements must be such that the elements meeting at a nodal point before deformation are subjected to the same displacements at that point. It should be mentioned here that, for a convergent solution,
the expression for element displacements needs to be of such a form that no gaps or overlapping take place along the sides of the elements.

(c) Force-Deflection Relationship: the relationship between internal forces and internal displacements required by the element properties must be obeyed.

A finite element formulation can be described as a variational problem and in elastic continuum analysis, it commonly involves minimisation of one of the two following quantities:

(i) total potential energy \((= (U + W))\),

(ii) total complementary potential energy \((= (U + W)_C)\)

where \(U\) and \(W\) stand for strain energy and external work done respectively.

In the first case \(U\) is given by

\[
U = \frac{1}{2} K_{ij} q_i q_j \quad \text{ ..................(11.1a)}
\]

where \(K_{ij}\) are the elements of the stiffness matrix and \(q_i, q_j\) are generalised nodal displacements, and the problem is described in terms of displacements of nodes.

This is generally known as the stiffness or displacement method\(^7\) and usually provides a simpler formulation of problems and easier computation for complex structures than other methods.

In the second case \(U\) is given by

\[
U_C = \frac{1}{2} S_{ij} p_i p_j \quad \text{ ..................(11.1b)}
\]
where $S_{ij}$ are the elements of the flexibility matrix and $p_i, p_j$ are nodal forces, and a problem is described in terms of nodal forces.

This is generally known as the force or flexibility method.

However, a method known as the hybrid or mixed method uses the principle of minimum total complementary energy to derive the stiffness matrix, and a problem is described in terms of generalised nodal displacements by this method.

In this work, the displacement and the hybrid methods were used, and a brief account of each of these is given below.

11.1a The Displacement Method

In this method, a displacement field is assumed for each element used to represent a structure. The unknown parameters of this field are then expressed in terms of the nodal displacements which then become the generalised degrees of freedom of the element.

The total potential energy, $\Pi$, is given by

$$\Pi = \int_V U_o \, d\text{vol} - \Omega \quad \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots (11.1c)$$

where $U_o$ is the strain energy per unit volume and is given by

$$U_o = \frac{1}{2} \sigma_{ij} \varepsilon_{ij} \quad \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots (11.1d)$$

$\sigma_{ij}$ and $\varepsilon_{ij}$ being the stress and strain components respectively, and $\Omega$ is the total external work done.
Now substituting the displacement field in terms of nodal displacements in $U_0$ and differentiating with respect to these nodal displacements we get

$$\frac{3}{\delta_e} \left( \int_V U_0 \, d\text{vol} \right) = \left[ K_e \right] \{\delta_e\} \quad \ldots \ldots \quad (11.1e)$$

where $[K_e]$ is the stiffness matrix for one element and $\{\delta_e\}$ are the nodal displacements of the element.

Now considering the structure as a whole:

$$\frac{3}{\delta_e} \left( \int_V U_0 \, d\text{vol} \right) = \text{no. of el's} \sum_{i=1}^{n} \left[ K_e \right] \{\delta_e\}$$

$$= \left[ K \right] \{d\} \quad \ldots \ldots \quad (11.1f)$$

where $[K]$ is the stiffness matrix for the whole structure and is obtained by superimposing stiffnesses of elements connecting to one node, and $\{d\}$ is the set of generalised displacements chosen for the analysis of the whole structure.

Then assuming that all external loads are acting at nodal points,

$$\Omega = - \{F\}^T \{d\} \quad \ldots \ldots \quad (11.1g)$$

where $d$ are the generalised displacements of the whole system, and $\{F\}$ are the nodal forces corresponding to $\{d\}$.

Then successive differentiation with respect to each of the generalised coordinates of the structure gives
Thus from the principle of stationary total potential energy we get

\[ \frac{\partial \pi}{\partial \{d\}} = 0 = [K]\{d\} - \{F\} \quad \text{(11.1i)} \]

so that the analysis of the structure is dependent on the solution of simultaneous linear equations:

\[ [K]\{d\} = \{F\} \quad \text{(11.1j)} \]

Because of positive definiteness of strain energy\textsuperscript{11} a unique solution is assured. As the stresses and strains within an element have previously been expressed in terms of displacements of its nodes, they may be calculated by direct substitution after solution of the above equations.

This method has been used in the solution of a very wide range of problems\textsuperscript{7}.

11.1b The Mixed Method

This approach was first used by Pian\textsuperscript{19} and subsequently by others\textsuperscript{15,20,21} for solving various continuum mechanics problems.

In this method an equilibrating stress field is assumed within an element and this is used, in conjunction with an assumed displacement field along the boundary, to develop the total complementary potential energy within an element. Then using the principle of stationary total complementary potential energy this method evaluates the element stiffness
matrix \([K_e]\) so that again a system is represented by a set of linear equations given by

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

Now the total complementary potential energy is given by

\[
\pi_C = U - \int u_i s_i \, ds \quad \ldots \ldots \ldots \ldots (11.1k)
\]

where \(U\) is the strain energy (for linearly elastic solid strain energy = complementary strain energy), \(u_i\) are surface displacements and \(s_i\) are the surface traction components.

The surface tractions can be given by

\[
s_i = \sigma_{ij} n_j \quad \ldots \ldots \ldots \ldots (11.1m)
\]

where \(\sigma_{ij}\) are the stress components at the surface and \(n_j\) are the direction cosines of the surface normal.

Now a stress field is assumed within an element such that

\[
\{\sigma\} = [P] \{\beta\} \quad \ldots \ldots \ldots \ldots (11.1n)
\]

where \([P]\) are functions of coordinates according to which the stress components are given, and \(\{\beta\}\) are a set of 'm' undetermined coefficients.

If the stress-strain relationship is given by

\[
\{\varepsilon\} = [N] \{\sigma\} \quad \ldots \ldots \ldots \ldots (11.1p)
\]

where \(\{\varepsilon\}\) are the strain components and \([N]\) is called the elasticity matrix, then

\[
U = \frac{1}{2} \int_V \{\sigma\}^T [N] \{\sigma\} \, dv \quad \ldots \ldots \ldots \ldots (11.1q)
\]
or
\[ U = \{\beta\}^T [H] \{\beta\} \] ..........(11.1r)

where
\[ [H] = \int_V [P]^T [N] [P] \, dv \] ..........(11.1s)

The components of boundary displacements, \{u\}, can be given by
\[ \{u\} = [L] \{q\} \] ..........(11.1t)

where \{q\} are a set of 'n' generalised nodal displacements and [L] describes the relationships between \{u\} and \{q\} in terms of the coordinates and on the boundary.

Now substituting equation 11.1n in equation 11.1m, the surface tractions can be given by
\[ \{S\} = [K] \{\beta\} \] ..........(11.1u)

Then the total complementary potential energy can be given by
\[ \pi_C = \{\beta\}^T [H] \{\beta\} - \{\beta\}^T [T] \{q\} \] ..........(11.1v)

where
\[ [T] = \int_S [R]^T [L] \, ds \] ..........(11.1w)

Now using the principle of minimum complementary total potential energy, i.e.
\[ \frac{\partial \pi_C}{\partial \{\beta\}} = 0, \quad [H] \{\beta\} = [T] \{q\} \] ..........(11.1x)

and then
\[ \{\beta\} = [H]^{-1} [T] \{q\} \] ..........(11.1y)
Now from equation 11.1q and equation 11.1y

\[ U = \frac{1}{2} \{q\}^t [T]^t [H]^{-1} [T] \{q\} \]

\[ = \frac{1}{2} \{q\}^t [K_e] \{q\} \]

\[ \ldots \ldots \ldots (11.1z) \]

and thus the element stiffness matrix for generalised nodal displacements, \( \{q\} \), is given by

\[ [K]_e = [T]^t [H]^{-1} [T] \]

\[ \ldots \ldots \ldots (11.1ab) \]

Then superimposing stiffness contributions from all elements to the common nodes, the overall stiffness matrix, \([K]\), can be obtained. Again a system is described by a set of linear equations:

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

The number of \( \beta \)-coefficients can be increased for greater accuracy, but computational difficulties and time usually put a practical upper limit on their number.

Although there is no upper limit, there is, however, a lower limit in the number of these coefficients. If \( r(K) \) gives the rank of the matrix \([K]\), then

\[ r(K) = \min(r([T]^t), r([H]^{-1}), r([T])) \]

\[ = \min(r([T]^t), r([H])) \]

\[ \ldots \ldots \ldots (11.1ac) \]

since \( r([T]^t) = r([T]) \)

and \( r(H) = r([H]^{-1}) \)

If for a given element type, the number of rigid body displacements is \( m \) and the total number of degrees of freedom is \( n \) then
\[ r(K) \geq n - m \]

But if the number of \( \beta \)-coefficients is less than \( n \) then

\[ \min(r([T]), r([H])) = \text{the number of } \beta \text{-coefficients,} \]

so that the number of \( \beta \)-coefficients is to be \( \geq n - m \).

However this rule is not absolute and in an extended system the number of independent equations may be enough to give an overall solution\(^{15} \), in spite of violating this rule.

11.2 The Loughborough University Finite Element System

Various departments of this University have been actively engaged in the finite element research for a number of years. At the time the work described in this thesis was started, a system program existed for static elastic analysis although this was not fully operational. An account of the existing system is given below.

The system is divided in the following independent units:

(a) **Program No 1**

This program reads data (coordinates of nodes, specifications of element types and nodes of each element, constraints-only zero displacements, and nodal forces). It performs several checks on data, calculates the half bandwidth, and writes a record on a magnetic tape (MT1) with the job name, number of nodes, number of degrees of freedom per node, number of elements and number of dimensions of the nodal coordinates.

It then puts the coordinates of nodes and data regarding
elements on MT1. The load vector(s) and an array, containing the rows of the stiffness matrix and the load vector(s) to be constrained, are set up and transferred to MT1.

Another magnetic tape (MT2) at the end of this stage carries only one record stating the job name, the number of nodes, the number of elements, the number of degrees of freedom per node, the number of dimensions of nodal coordinates, the number of elements, and half-bandwidth.

(b) Program No 2

This program uses the two magnetic tapes storing data for a particular job. It reads data regarding each element from MT1, and then calls the appropriate subroutine to calculate the element stiffness matrix. During the calculation of the stiffness matrix of each element, the entire stiffness matrix is kept on a disc backing store (using ICL subroutines UTD1, UTD2, UTD5 and UTD6). Each time the appropriate part of the whole stiffness matrix is brought to core, the stiffness contribution from the element is added to appropriate rows of the overall stiffness matrix and then this part of the overall stiffness matrix is transferred back to disc store again. At the same time, the element stiffness matrix and other necessary data regarding the element are written onto MT2.

When the stiffness matrices of all the elements have been calculated, the overall stiffness matrix is read back
from disc store and is written on MT2.

(c) **Program No 3**

This program uses two magnetic tapes and the disc backing store described above.

It first reads the numbers of rows of the stiffness matrix to be constrained from MT1. Then it reads the unconstrained stiffness matrix from MT2, and transfers it to the disc backing store. The overall stiffness matrix is then read from the disc store row by row and the appropriate elements of each row are treated. Then the constrained stiffness matrix is written on MT2.

Then the constrained stiffness matrix is triangularised by the Cholesky decomposition process. As the stiffness matrix \([K]\) is real, symmetric and positive definite,

\[
[K] = [L] [L]^T
\]

............... (11.2a)

where \([L]\) is a lower triangular matrix. This is done by reading the constrained stiffness matrix row by row from MT2 and storing the rows of \([L]\) in the disc backing store.

At the end of the operation, the lower triangular matrix, \([L]\), is stored on MT2.

(d) **Program No 4**

This program reads the array containing the numbers of elements of the load vector(s) that are to be constrained and the load vector(s) from MT1, and constrains the load vector(s).
It then reads the lower triangular matrix, formed by the decomposition of the overall stiffness matrix, from MT2 and transfers it to the disc backing store.

Then by back-substitution the generalised displacement vector(s) is/are calculated and written on MT2 as well as on a disc serial file. Then the nodal displacements are output on the line printer.

(e) Program No 5

This program reads the displacement vector(s) from the disc backing store, selects the nodal displacements of each element for different loading cases, reads the element stiffness or stress matrix and other data prescribing different element properties from MT2 and MT1, and then calculates and outputs the nodal forces and stresses within the element for different loading cases.

The division of the system program into several segments as above allows analysis of structures with the minimum of computer storage space. With the use of overlay facilities, the core space requirement of all of the constituent programs is less than 20K for a maximum capacity of half-bandwidth x No. of rows = 25,000. Also for a large structure requiring too large an amount of computer time to be available at any one time for a complete analysis, any one of the constituent programs may be run at a time. The most time-consuming programs are the ones calculating and assembling the stiffness matrix, and decomposing the stiffness matrix. So the division
of the system into two main segments, the first containing the stiffness matrix generation program and second containing the Cholesky decomposition program allows reasonably large structures to be analysed. Another advantage of segmentation is that the time lost due to unexpected failures due to faults in the computer accessories is minimal.

11.3 The System used by the Writer

At the time the writer started his work, the original system described above was not operative because the computer, for which it was developed, was replaced shortly before by the present computer. Although the change of computer did not involve many changes in the programs, several inconsistencies between the two computers gave rise to some incompatibilities between the programs and the new computer. Some of these took a great deal of effort and time to trace and rectify, but do not merit their presentation here.

However, some modifications of the above programs were carried out to improve their efficiency and an account of a few of these are given below:

11.3a Modifications made in the existing Finite Element System Program

Although the above finite element system program was fairly general, the use of the disc backing store involving ICL programs UTD1, UTD2, UTD5 and UTD6 imposes some limitations on the program. First of all, they allow only 50K of disc space to be used and this, in practical terms, means that the
stiffness matrix must be such that half-bandwidth × the total no. of rows is to be less than or equal to 25,000. This imposes a limitation on the accurate analysis of large structures. Also UTD programs occupy some core space which could be utilised for some other purposes. Moreover, because the readings and writings with UTD's are done in blocks, some transfer time and disc storage space is likely to be wasted.

To overcome these difficulties the UTD programs have been withdrawn from the entire system and have been replaced by direct access disc files. This allows special disc files to be used without any practical limitation on the disc storage space required. This means that considerably larger structures can now be analysed with greater accuracy.

In the existing system all nodes of all elements have to have the same number of degrees of freedom. This means that structures consisting of two element types with different number of degrees of freedom per node meeting at a node on a common boundary cannot be analysed in spite of physical compatibility between these two element types along the common boundary.

In the present system facilities have been introduced whereby pseudo-degrees of freedom may be assigned to elements having lower number of degrees of freedom per node than some other elements to bring the size of this element stiffness matrix to that of the element with maximum degrees of freedom per node. Now the problem of assembly of the stiffness matrix is eliminated and the constraining of the redundant degrees of
freedom leads to the appropriate number of independent equations.

The original system also does not have any facility whereby an element with different nodes with different degrees of freedom may be used. This requirement poses some programming problems which have been overcome in the present system. Thus the present system offers facilities for incorporating elements with different nodes having different degrees of freedom.

In the present system a subroutine has been incorporated in order to calculate the mass acceleration at each element for a given rotational speed, distribute the equivalent force to its nodes, resolve the nodal forces in the directions of the global coordinate axes, and superimpose these resolved forces to form an overall force vector for the whole system.

In many computer installations, like those in universities, there is a limitation on the time allowed for one program at a time. Thus, although the analysis of a structure requiring substantially more time for running one or more of the constituent programs than is allowed at any one time, may be economic, but cannot be performed in the original system. Facilities have been introduced in the present system whereby the calculations performed up to a specified point within a constituent program can now be transferred on to magnetic tapes for a further run when time becomes available. This process implies that the output of final results can take as many individual runs as desired. When comparatively accurate results are required this process can reduce the need for analysis involving treatment of a segment of a structure with fine mesh, the rest of the structure being represented by a fairly coarse mesh, and further runs with
fine mesh treatment elsewhere. The latter method consumes more data preparation and computation time for a complete analysis and should be avoided if possible.

A flow chart of the present system is shown in Chapter 11.5. It includes the modified versions of the original programs: N307 (Program No 1), N308 (Program No 2), N310 (Program No 3), N311 (Program No 4), N313 (Program No 5), as well as the two new programs to accomplish the constraint matrix operations: N309 and N312. The programs N309 and N312 are described below.

To deal with cases requiring the use of the constraint matrices (Chapter 3), the programs N309 and N312 have been introduced. The use of these two programs eliminate the need for introducing pseudo degrees of freedom except in special cases. As the use of pseudo degrees of freedom may mean substantial increase of computation, in order to reduce this wastage, instead of bringing the number of degrees of freedom per node, the data presentation now indicates whether elements with different degrees of freedom per node are to be used and, if so, the subsequent computation is performed on this basis so that the stiffness matrix is generated in blocks involving one type of element only. The two new programs are used to combine these blocks for the integrated stiffness matrix and the load vector leading to an overall solution of displacements and stresses.

Program N309: This program is based on the contents of Chapter 3. Basically this program performs the following
operations:

(1) \[ [K'] = [C]^t[K][C] \]

where \([K']\) is the new integrated stiffness matrix, \([K]\) the original stiffness matrix formed of disjointed blocks, and \([C]\) is the constraint matrix (see Chapters 3 and 4 for detailed description) and, in general of the form

\[
[C] = \begin{bmatrix}
[I_1] & [0] & [0] \\
\hline
[0] & [C_t] & [0] \\
[0] & [0] & [I_2]
\end{bmatrix}
\]

where \([I_1]\) and \([I_2]\) are identity matrices.

(2) \{F'\} = [C]^t\{F\}

where \{F\} is the force vector of the integrated system and \{F\} is the original force vector corresponding to \([K]\).

Taking advantage of the symmetric properties of \([K]\) and \([C]^t[K][C]\) and using direct access disc files this program has been written in such a way that fairly large structures can be handled economically.

In some particular cases \([C_t]\) is generated automatically from simple data input. At present, automatic generation of \([C_t]\) is possible in the following cases: matching shell elements with axisymmetric elements, using axisymmetric and plane elements together, and imposing boundary conditions by equalising nodal displacements on one part of a boundary with
nodal displacements on another corresponding part of the boundary in terms of cylindrical coordinates. At the end of this program the integrated stiffness matrix \([K']\) is written on MT2, and the constraint matrix \([C_t]\), and the load vector, \(\{F'\}\), on MT1.

A flow chart of this program is given in Chapter 11.5b.

**Program N312:** This program is used if the program N309 has been used before on the same job. It performs the following operation

\[
\{u\} = [C]^t\{u'\}
\]

where \(\{u\}\) is the displacement vector of individual nodal displacements and \(\{u'\}\) is the displacement vector of the constrained system containing only the independent displacements.

\([C_t]\) is read from MT1 and the above operation is performed to get \(\{u\}\). It then prints the nodal displacements on the line printer and the displacement vector \(\{u\}\) on a disc file for subsequent use by the program N213.

11.4 Element Types used in this Report for the Analysis of Test Cases and Rotors

11.4a The Plane Stress Element

This is a simple constant strain triangle. The assumed displacement functions are given by

\[
\begin{align*}
    u &= a_1 + a_2x + a_3y \\
    v &= a_4 + a_5x + a_6y
\end{align*}
\] ........................ (11.4a)
The a's are expressed in terms of nodal displacements, 
\( \{\delta_e\} = \{u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3\}^t \).

Now \( U_e \) = strain energy in the element
\[
U_e = \frac{t}{2} \int \{\sigma\}^t \{\varepsilon\} dx \ dy
\]
(11.4b)

where \( t \) = thickness of the element

Now \( \{\sigma\} = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{bmatrix} = \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} \)
(11.4c)

and

\( \{\varepsilon\} = \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} \)
(11.4d)

Substituting from equations 11.4c and 11.4d in 11.4b and expressing \( \{\varepsilon\} \) in terms of nodal displacements, \( \{\delta_e\} \),
the element strain energy can be given by

\[ U_e = \frac{1}{2} \{ \delta_e \}^t \begin{bmatrix} K_e \end{bmatrix} \{ \delta_e \} \]  \hspace{1cm} (11.4e)

where \([K_e]\) is the element stiffness matrix.

11.4b The Axisymmetric Solid Element

The formulation in this case is also based on the displacement method.

This is again a two-dimensional problem. The chosen displacement functions are

\[ \begin{align*}
  u &= a_1 + a_2 r + a_3 z \\
  v &= a_4 + a_5 r + a_6 z
\end{align*} \hspace{1cm} (11.4f) \]

\( \{ \delta_e \} = \{ u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \}^t \).

Now \[ U_e = \pi \int \{ \sigma \}^t \{ \varepsilon \} r \, dr \, dz \]
\[ \sigma = \begin{bmatrix} \sigma_z \\ \sigma_r \\ \sigma_\theta \\ \tau_{rz} \end{bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \begin{bmatrix} \varepsilon_z \\ \varepsilon_r \\ \varepsilon_\theta \\ \gamma_{rz} \end{bmatrix} \] (11.4g)

and

\[ \{\varepsilon\} = \begin{bmatrix} \varepsilon_z \\ \varepsilon_r \\ \varepsilon_\theta \\ \gamma_{rz} \end{bmatrix} = \begin{bmatrix} 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial r} & 0 \\ \frac{1}{r} & 0 \\ \frac{\partial}{\partial z} & \frac{\partial}{\partial r} \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix} \] \hspace{1cm} (11.4h)

Then again using the same basic principles mentioned before

\[ U_e = \frac{1}{2} \{\varepsilon_e\}^T [K_e] \{\varepsilon_e\} \]

where \([K_e]\) is the element stiffness matrix.

11.4c The Shell Element

This is a flat plate element\(^{15}\) combining the stiffness matrices due to a plane stress formulation and a plate bending formulation.

Both the plane and the plate bending stiffness matrices are based on the mixed method. A brief description of each of the above two formulations is given below:
i) **The Plane Stress Stiffness Matrix**

This is a polygonal element with the following degrees of freedom per node:

\[
u, v, \theta (= \frac{1}{2} (\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}))\text{, and } e (= \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}).
\]

Now

\[
\sigma_x = \frac{\partial^2 \phi}{\partial y^2}, \quad \sigma_y = \frac{\partial^2 \phi}{\partial x^2}, \quad \tau_{xy} = \frac{\partial^2 \phi}{\partial x \partial y},
\]

where \(\phi\) is the Airy stress function.

The following form is assumed for \(\phi\):

\[
\phi = A_1(y^2/2) + A_2(y^3/6) + A_3(x^2/2) + A_4(x^3/6) + A_5(xy)
+ A_6(xy^2/2) + A_7(x^2y/2) + A_8(y^4/12) + A_9(x^4/12) + A_{10}(x^2y^2/2)
+ A_{11}(x^3y/6) + A_{12}(xy^3/6) + A_{13}(y^5/20) + A_{14}(xy^4/12)
+ A_{15}(x^2y^3/2) + A_{16}(x^3y^2/2) + A_{17}(x^4y/12) + A_{18}(x^5/20)
\]

However in order to satisfy the bi-harmonic equations

\[(v^4 \phi = 0)\text{, it is found that.}\]
Therefore the number of independent stress coefficients is 15. From the above the \([P]\) matrix is given by

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy}
\end{bmatrix} =
\begin{bmatrix}
1 & y & 0 & 0 & 0 & x & 0 & \left(y^2 - \frac{x^2}{2}\right) & -\frac{1}{2}x^2 \\
0 & 0 & 1 & x & 0 & 0 & y & -\frac{y^2}{2} & \left(x^2 - \frac{1}{2}y^2\right) \\
0 & 0 & 0 & 0 & 1 & -y & -x & xy & xy
\end{bmatrix}
\]

Now from the Stress-Strain relationship of plane stress problems the elasticity matrix is given by

\[
\begin{bmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy}
\end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}
\]

or

\[
\{\varepsilon\} = [N]\{\sigma\}
\]

The H-matrix (Chapter 11.1b) is now obtained from the equations 11.1s which gives

\[
[H] = \int_V [P]^T [N] [P] dV
\]
Again referring to Chapter 11.1b, the interpolation matrix, $L$, is obtained by assuming that the normal and in-plane boundary displacements are given by

$$
u' = a_1 s^3 + a_2 s^2 + a_3 s + a_4$$

and $$
u' = a_5 s^3 + a_6 s^2 + a_7 s + a_8$$

where $s = \frac{x'}{t}$ (see Figure 11.5)

The $a$'s are expressed in terms of generalised nodal displacements by using a transformation matrix, and the matrix, $L$, is obtained.

Then the boundary traction matrix, $R$ (see equation 11.1u), is obtained by putting the equations of the boundaries into the $P$-matrix and using a transformation matrix, so that the $R$-matrix gives the normal and shear forces along the boundaries.

Now the $T$-matrix is obtained from the equation 11.1w which gives

$$[T] = \int_{V} [R]^T[L] ds$$

and so the element stiffness matrix, $K_e$, is obtained from

$$[K_e] = [T]^T[H]^{-1}[T]$$

ii) The Plate Bending Stiffness Matrix

In this case the stress functions chosen are as follows:
\[
\begin{align*}
\sigma_x &= \frac{8z}{t} (\beta_1 + \beta_2 x + \beta_3 y) \\
\sigma_y &= \frac{8z}{t} (\beta_4 + \beta_5 x + \beta_6 y) \\
\tau_{xy} &= \frac{8z}{t} (\beta_7 + \beta_8 x + \beta_9 y) \\
\tau_{xz} &= t (1 - \frac{4z^2}{t^2}) (\beta_2 + \beta_3) \\
\tau_{yz} &= t (1 - \frac{4z^2}{t^2}) (\beta_6 + \beta_8)
\end{align*}
\] 

\text{where } t = \text{ thickness of the plate.}

\text{Figure 11.6}

The above stress fields satisfy the following equilibrium equations:

\[
\begin{align*}
\frac{\partial Q_x}{\partial x} + \frac{\partial Q_y}{\partial y} &= 0 \\
\frac{\partial M_{xy}}{\partial x} + \frac{\partial M_y}{\partial y} - Q_y &= 0 \\
\frac{\partial M_{xy}}{\partial y} + \frac{\partial M_x}{\partial x} - Q_x &= 0
\end{align*}
\] 

\text{..........(11.4n)}
where

\[
\begin{align*}
M_x &= \int_{-t/2}^{t/2} \sigma_x z \, dy, \\
M_y &= \int_{-t/2}^{t/2} \sigma_y z \, dz, \\
M_{xy} &= \int_{-t/2}^{t/2} \tau_{xy} z \, dz, \\
Q_x &= \int_{-t/2}^{t/2} \tau_{yz} \, dz, \\
Q_y &= \int_{-t/2}^{t/2} \tau_{zy} \, dz.
\end{align*}
\]

\[\ldots \ldots \ldots \ldots (11.4p)\]

Now the P-matrix is given by

\[
\begin{bmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy} \\
\tau_z \\
\tau_{yz}
\end{bmatrix} = 
\begin{bmatrix}
A & Ax & Ay & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & A & Ax & Ay & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & A & Ax \\
0 & 0 & B & 0 & 0 & 0 & 0 & B \\
0 & 0 & 0 & 0 & B & 0 & B & 0
\end{bmatrix}\begin{bmatrix}
\beta_1 \\
\beta_2 \\
\beta_3 \\
\beta_4 \\
\beta_5 \\
\beta_6 \\
\beta_7 \\
\beta_8 \\
\beta_9
\end{bmatrix}
\]

\[\ldots \ldots \ldots \ldots (11.4q)\]

\[
= [P]\{\beta\}
\]

where \( A = \frac{8z}{t} \), and \( B = t \times (1 - \frac{4z^2}{t^2}) \).

The elasticity matrix, \( N \), is given by
\[
\begin{pmatrix}
\varepsilon_x \\
\varepsilon_y \\
\gamma_{xy} \\
\gamma_{xz} \\
\gamma_{yz}
\end{pmatrix} = \frac{1}{E} \begin{pmatrix}
1 & -\nu & 0 & 0 & 0 \\
-\nu & 1 & 0 & 0 & 0 \\
0 & 0 & \bar{v} & 0 & 0 \\
0 & 0 & 0 & \bar{v} & 0 \\
0 & 0 & 0 & 0 & \bar{v}
\end{pmatrix} \begin{pmatrix}
\sigma_x \\
\sigma_y \\
\tau_{xy} \\
\tau_{xz} \\
\tau_{yz}
\end{pmatrix}
\]

where \( \bar{v} = 2(1 + \nu) \),
or \( \begin{bmatrix} \varepsilon \end{bmatrix} = [N] \begin{bmatrix} \sigma \end{bmatrix} \)

Then the \( H \)-matrix is obtained by

\[
[H] = \int_V \begin{bmatrix} p \end{bmatrix}^T [N] [p] dv
\]

Assuming a cubic variation of the vertical displacement, \( w \), and a linear variation of rotation, \( \theta_t \) (see Figure 11.6), the edge displacements \( (w, \theta_t, \theta_n) \) are expressed in terms of the nodal displacements of a side. So the interpolation matrix, \( L \), is obtained such that

\[
\begin{bmatrix} w \\ \theta_t \\ \theta_n \end{bmatrix} = [L] \{q\}
\]

where \( \{q\} \) are the generalised nodal displacements.

Then the components of the edge tractions, \( (Q, M_t, M_n) \) in the directions of \( w, \theta_t \) and \( \theta_n \) are obtained from \( Q_x, Q_y, M_x, M_y \) and \( M_{xy} \) at the boundaries by means of a transformation matrix and so the \( R \)-matrix is obtained (see equation 11.1u).

Now the \( T \)-matrix is obtained from

\[
[T] = \int_V \begin{bmatrix} K \end{bmatrix}^T [L] ds
\]

and thus the element
stiffness matrix, $[K_e]$, is obtained from

$$[K_e] = [T]^t[H]^{-1}[T]$$

iii) Combining the above Two Element Types into the Shell Element

A thin shell may be defined as one obeying the following conditions:

(a) no shear stress develops between the inside and the outside surfaces,

(b) direct and shear stress in the plane of the element vary linearly across the thickness of the element,

(c) out of plane shear stresses vary parabolically across the thickness, with zero values at the surfaces and the maximum value at the midplane.

The plane stress element and the plate bending elements mentioned above obey the above conditions and therefore a combination of these two element formulations can be regarded as a suitable description of a shell.

Thus a shell element formulation is obtained with the following degrees of freedom per node: $u$, $v$, $w$, $\theta_x$, $\theta_y$, $\theta_z$ and $\varepsilon$ (see Chapter 11.4c(i)).

However if a shell is described as an assembly of flat plates, each of these plates generally lie on different planes. To facilitate computation each element stiffness matrix is first developed in terms of a set of local coordinate axes lying on the plane of the element. Then this element
stiffness matrix is expressed in terms of the global coordinate system with the help of a transformation matrix.

Let \( \{q\} = \{u, v, w, \theta_x, \theta_y, \theta_z, e\}^t \) be the global degrees of freedom per node and \( \{q\} = \{u', v', e', \theta'_z, w', \theta'_x, \theta'_y\}^t \) be the local degrees of freedom per node for one element.

If

\[
\begin{bmatrix}
u' \\
v' \\
w'
\end{bmatrix} = \begin{bmatrix} l_1 & l_2 & l_3 \\ m_1 & m_2 & m_3 \\ n_1 & n_2 & n_3 \end{bmatrix} \begin{bmatrix} u \\
v \\
w \end{bmatrix} = [L'] \begin{bmatrix} u \\
v \\
w \end{bmatrix}
\]

where \([L']\) is the direction cosine matrix, then

\[
\begin{bmatrix}
u' \\
v' \\
w' \\
\theta'_z \\
\theta'_x \\
\theta'_y
\end{bmatrix} = \begin{bmatrix}
l_1 & l_2 & l_3 & 0 & 0 & 0 & 0 \\
m_1 & m_2 & m_3 & 0 & 0 & 0 & 0 \\
n_1 & n_2 & n_3 & 0 & 0 & 0 & 1 \\
m_3l_2 - l_3m_2 & m_1l_3 - l_1m_3 & m_2l_1 - l_2m_1 & 0 & \theta_x \\
n_3m_2 - n_2m_3 & n_1m_3 - n_3m_1 & n_2m_1 - n_1m_2 & 0 & \theta_y \\
m_3n_2 - n_2n_3 & n_1n_3 - n_3n_1 & n_2n_1 - n_1n_2 & 0 & \theta_z
\end{bmatrix}
\]

Thus \( \{q'\} = [C]\{q\} \)

Now if \([K'_1]\) and \([K'_2]\) are the plane and the bending stiffness matrices of an element respectively in terms of a local coordinate system then

\[
[K'] = \begin{bmatrix} [K'_1] & [0] \\
[0] & [K'_2] \end{bmatrix}
\]

\ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \li
and

\[ [K_e] = [\mathcal{C}]^t [K_e] [\mathcal{C}] \]  \hspace{1cm} (11.4v)

where \([K_e]\) is the element stiffness matrix in terms of the local coordinate system and \([K_e]\) is that in terms of the global coordinate system.
11.5 Flow Charts

11.5a Flow Chart of the Present System Program

**PROGRAM N307**
It reads data from cards, performs a few checks on data, calculates the load vector and writes data on two magnetic tapes.

**PROGRAM N308**
It calculates the element stiffness matrices and assembles them into the overall stiffness matrix.

**PROGRAM N309**
(Optional)
It performs the constraint matrix operations: 

\[ [K'] = [C]^T[X][C] \]

and

\[ [F'] = [C]^T[F] \]

(see detailed flow chart below).

**PROGRAM N310**
This program constrains the overall stiffness matrix rows which are to be directly (zero displacement) constrained. It then performs the triangular decomposition on the overall stiffness matrix, X, such that

\[ X = [L][L]^T \]

(see Chapter 11.2).

**PROGRAM N311**
It calculates the displacement vector by back substitution and lists the nodal displacements.

**PROGRAM N312**
(Optional)
If Program N309 has been used, then this program is used to calculate and list the original nodal displacements \((u) = [C][u']\), (see Chapter 11.2).

**PROGRAM N313**
This program calculates and prints the stresses in each element at specified points.
Program N309

Read data from cards

Read data from MTs

Is the constraint matrix \( [C_t] \) to be generated automatically?

No

Read the constraint matrix from cards in rows

Yes

Generate the constraint matrix in rows

Is the constraint matrix too big for core?

No

Write on MT1 the constraint matrix, \( [C_t] \), column by column

Yes

Write a direct access disc file, the constraint matrix, each record containing a column

Read columns of the constraint matrix from the direct access disc file

Apply the constraining operations on the load vector \( ([F']) = [C]^{-1}(F) \) and write the constrained load vector on MT1

Read from MT2 the rows of \([A_1]\)

Is the bandwidth likely to change?

No

Yes

Write on MT3 the rows of \([A_1]\)

This program performs the operations described in Chapter 3.1
Read the stiffness matrix rows from MT2

Is core large enough to store \([A2]\)?

Yes

Form \([A2]\) matrix

No

Write elements of \([A2]\) on Direct Access disc file so that each record contains a column of \([A2]\)

Write the rows of the lower triangle of \([A3]\) on MT1

Is core large enough to store \([A2]\)?

Yes

Read columns of \([C_t]\) from MT1 and \([A3]\) from MT2

Form rows of \([C_t]^T[A2]\) and write on MT3 and check for bandwidth

No

Read columns of \([C_t]\) from MT1 and \([A2]\) from MT2

Read rows of the lower triangle of \([A3]\) from MT1

Is core large enough to store \([A3]\)?

Yes

Read the lower triangle of \([A3]\) from MT1 and complete \([A3]\)

No

Read segments of \([A3]\) from MT1 and write on Direct Access disc file so that each record contains a row or a column

Read from MT1 a column of \([C_t]\)

Read from direct access disc files columns of \([A3]\)

Form rows of \([C_t]^T[A3]\) and write on direct access disc file
Read from MT1 the columns of $[C_r]$.

Form rows of $[C_r]^T[A_3]$ and write on direct access disc file.


Read columns of $[C_r]$ from MT1; form rows of $[C_r][A_3][C_r]$.

Check for bandwidth.

Is $[C_r]$ small enough to be placed in core 2?

Read $[C_r]$ from MT1.


Form rows of $[A_5][C_r]$.

Check for bandwidth.

Write the rows of $[A_4]$, $[A_5][C_r]$ and $[A_6]$ consecutively on MT1.

Was core large enough to store $[C_r]$?

Yes

Read back from MT1 and MT3 the rows of the lower triangle of the constrained stiffness matrix and write on MT2.

END
11.6 The Finite Element Meshes used in Chapters 6, 7, 8 and 9
Figure 6.2: The Finite Element Mesh for the Radial Flow Rotor
Strain Gauge positions are marked Θ

Figure 7.1b: The Finite Element Mesh in Part 4 of Rotor Type BD1
Figure 7.1c: The Finite Element Meshes in Parts 1 and 6 of Rotor Type BD1.
Strain Gauge positions are marked.

Figure 7.1d: The Finite Element Mesh in Part 5 of Rotor Type BD1
Strain Gauge positions are marked.

Figure 7.1c: The Finite Element Mesh in Part 2 of Rotor Type BD1
Figure 7.2c: The Finite Element Mesh in Part 1 of the Rotor Type JH1
Figure 7.2d: The Finite Element Mesh in Parts 2 and 3 of Rotor Type JH1
Figure 7.2e: The Finite Element Mesh in part 4 of the Rotor Type JH1
Figure 7.2f: The Finite Element Mesh in part 5 for the Rotor Type JH1
Figure 8.4: The Finite Element Meshes for Testing the Cylindrical Element (Chapter 8)
Figure 9.6: The Finite Element Mesh for the Plate Bending Problem
11.7 Graphs presenting results obtained in Chapters 6, 7 and 8
Graph 6.1: Radial Stresses at the Rear of the Backplate (Beneath Blades) of the Radial Rotor (Figure 6.1)
Graph 6.2: Radial Stresses at the Front of the Backplate (Beneath Blades) of the Radial Rotor (Figure 6.1)
Graph 6.3: Circumferential Stresses at the Rear of the Backplate (Beneath Blades) of the Radial Rotor (Figure 6.1)
Graph 6.4: Circumferential Stresses at the Front of the Backplate (Beneath Blades) of the Radial Rotor (Figure 6.1)
Graph 6.5: Blade Stresses at the Root of the Radial Rotor (Figure 6.1)
Graph 6.6: Blade Stresses at the Leading Edge of the Radial Rotor (Figure 6.1)
"Equivalent Von Mises Stress" for the Inside Surface

Circumferential Stresses

The junction between the axisymmetric part and the shell part

x - outside
o - inside

Radial Stresses

Graph 7.1a: Stresses in the Front Shroud (Between Blades) of the Rotor Type BD1
(Figure 7.1a)
Graph 7.1b: Maximum Principal Stresses in Blade along 'YY' in the Rotor Type BD1
(Figure 7.1a)
Point of attachment of the front shroud with the blade

Graph 7.1c: Maximum Principal Stresses along 'XX' in Blade of the Rotor Type (Figure 7.1a)
Graph 7.1d: Maximum Principal Stresses in the Blade (at the Rear End of the Rotor, Type B/D)
Graph 7.1e: Maximum Principal Stresses in the Blade (at the front end) of the Rotor Type BD1 (Figure 7.1a)
Graph 7.1f: Radial Displacements along the line 'ZZ' in Rotor Type BD1 (Figure 7.1a)
Graph 7.1g: Measured Strain vs (R.P.M.)^2 for the Rotor Type BD1
Graph 7.2a: Circumferential Stresses in the Front Shroud 1" in front of Blades (along 'XX' in Figure 7.2a)
Graph 7.2b: Circumferential Stresses in the Front Shroud between Blades (along 'ZZ' in Figure 7.2a)
Graph 7.2c: Circumferential Stresses in the Front Shroud 1" behind Blades
(along 'YY' in Figure 7.2a)
Graph 7.2d: Normal Stresses on the Trailing Blade Surface between Shrouds (along 'BB' in Figure 7.2a)
Graph 7.2e: Normal Stresses in the Trailing Blade Surface near the Back Shroud (along 'CC' in Figure 7.2a)
Graph 7.2f: Longitudinal Stresses in the Blade Trailing Face near the Front Shroud (along 'AA' in Figure 7.2a)
Graph 7.2g: Normal Stresses in the Trailing Blade Surface near the Front Shroud (along 'AA' in Figure 7.2a)
Graph 7.2h: Longitudinal Stresses on the Trailing Blade Surface between Shrouds (along 'BB' in Figure 7.2a)
Graph 7.2i: Longitudinal Stresses in the Trailing Blade Surface near the Back Shroud (along 'CC' in Figure 7.2a)
Graph 7.2j: Normal Stresses on the Blade Leading Face near the Front Shroud (along 'AA' in Figure 7.2a)
Graph 7.2k: Longitudinal Stresses on the Blade Leading Face near the Front Shroud
Graph 7.22: Normal Stresses on the Blade Leading Face near the Back Shroud (along 'CC' in Figure 7.2a)
Graph 7.2m: Longitudinal Stresses on the Leading Blade Surface near the Back Shroud (along 'CC' in Figure 7.2a)
Graph 7.2n: Longitudinal Stresses on the Leading Blade Surface between Shrouds (along 'BB' in Figure 7.2a)
Graph 7.2p: Normal Stresses on the leading Blade Surface between Shrouds (along 'BB' in Figure 7.2a)
Graph 7.2q: Maximum Principal Stress in the Back Shroud in Front of the Blade (along 'XX' in Figure 7.2a)
Graph 7.2r: Maximum Principal Stress in the Back Shroud between Blades (along 'ZZ' in Figure 7.2a).
Graph 7.2s: Maximum Principal Stress in the Back Shroud behind the Blade (along 'YY' in Figure 7.2a)
Graph 8.4a: Vertical Displacement of a point, A, on the Periphery
Cylindrical Solid Element

Axisymmetric Element

Graph 8.4b: Radial Displacement of a Point, A, on the Periphery
11.8 The Test Rig for Strain Gauge Experiment on Rotor Type BD1
UNIVERSITY OF TECHNOLOGY.

LOUGHBOROUGH,

MECHANICAL ENGINEERING.
UNIVERSITY OF TECHNOLOGY
LOUGHBOROUGH
MECHANICAL ENGINEERING.
12. REFERENCES


