REPORT DOCUMENTATION PAGE

1. Recipient's Reference	2. Originator's Reference	3. Further Reference	4. Security Classification of Document
	AGARD-LS-147	ISBN 92-835-1536-6	UNCLASSIFIED

5. Originator Advisory Group for Aerospace Research and Development
North Atlantic Treaty Organization
7 rue Ancelle, 92200 Neuilly sur Seine, France

6. Title
PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN

7. Presented at

8. Author(s)/Editor(s)	9. Date
Various	August 1986

10. Author's/Editor's Address	11. Pages
Various	140

12. Distribution Statement This document is distributed in accordance with AGARD policies and regulations, which are outlined on the Outside Back Covers of all AGARD publications.

13. Keywords/Descriptors

Aircraft panels	Models
Aircraft	Design
Mathematical models	Structural analysis

14. Abstract

This Lecture Series will introduce the main aspects of finite element analysis and modelling with emphasis placed on the solution of practical design problems. An outline will be given of the broad principles of the finite element method with some emphasis on the limitations of the technique. This will be followed by an explanation of the modelling problems encountered in the analysis of real structures together with their resolution. The use of one of the FE systems will be included.

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

AGARD-LS-147

AGARD-LS-147

AGARD

ADVISORY GROUP FOR AEROSPACE RESEARCH & DEVELOPMENT, Paris.

7 RUE ANCELLE 92200 NEUILLY SUR SEINE FRANCE

AGARD LECTURE SERIES No.147

Practical Application of Finite Element Analysis to Aircraft Structural Design.

NORTH ATLANTIC TREATY ORGANIZATION

DISTRIBUTION AND AVAILABILITY
ON BACK COVER

AGARD-LS-147

NORTH ATLANTIC TREATY ORGANIZATION

ADVISORY GROUP FOR AEROSPACE RESEARCH AND DEVELOPMENT

(ORGANISATION DU TRAITE DE L'ATLANTIQUE NORD)

AGARD Lecture Series No.147

PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

THE MISSION OF AGARD

The mission of AGARD is to bring together the leading personalities of the NATO nations in the fields of science and technology relating to aerospace for the following purposes:

— Exchanging of scientific and technical information;

— Continuously stimulating advances in the aerospace sciences relevant to strengthening the common defence posture;

— Improving the co-operation among member nations in aerospace research and development;

— Providing scientific and technical advice and assistance to the Military Committee in the field of aerospace research and development (with particular regard to its military application);

— Rendering scientific and technical assistance, as requested, to other NATO bodies and to member nations in connection with research and development problems in the aerospace field;

— Providing assistance to member nations for the purpose of increasing their scientific and technical potential;

— Recommending effective ways for the member nations to use their research and development capabilities for the common benefit of the NATO community.

The highest authority within AGARD is the National Delegates Board consisting of officially appointed senior representatives from each member nation. The mission of AGARD is carried out through the Panels which are composed of experts appointed by the National Delegates, the Consultant and Exchange Programme and the Aerospace Applications Studies Programme. The results of AGARD work are reported to the member nations and the NATO Authorities through the AGARD series of publications of which this is one.

Participation in AGARD activities is by invitation only and is normally limited to citizens of the NATO nations.

The content of this publication has been reproduced
directly from material supplied by AGARD or the authors.

Published August 1986

ISBN 92-835-1536-6

Printed by Specialised Printing Services Limited
40 Chigwell Lane, Loughton, Essex IG10 3TZ

LIST OF SPEAKERS

Lecture Series Director: Dr A.J.Morris
Department of Aircraft Design
College of Aeronautics
Cranfield,
Bedford MK 43 0AL
United Kingdom

SPEAKERS

Mr I.C.Taig
British Aerospace PLC
Warton Division
Warton
Nr. Lytham St. Annes, Lancashire
United Kingdom

Dr V.B.Venkayya
Air Force Wright Aeronautical Lab.
Wright Patterson Air Force Base
Ohio 45433-6553
USA

CONTENTS

INTRODUCTION TO FINITE ELEMENT BASICS

A.J. MORRIS

Professor of Computational Structural Analysis

College of Aeronautics,
Cranfield Institute of Technology,
Cranfield,
Bedford.
MK43 OAL
U.K.

1. PHILOSOPHY

The task we are setting ourselves is that of solving a complex structural design problem which lies beyond the scope of classical closed form mathematical solution. If we have a digital computer at our disposal we would be well advised to employ it and, because of the nature of the digital process, we will require an approximate solution technique. In addition, we would want to choose our new technique to suit the special properties of the computer. We require, in fact, a solution technique which is numerically stable, easily programmed and can be adapted to a wide range of problem types without excessive interference by the user. From a structural viewpoint the finite element method provides the most satisfactory solution technique in this category.

The essence of the finite element method involves dividing the structure into a suitable number of small pieces called finite elements. The intersections of the sides of the elements occur at nodal points or nodes and the interfaces between elements are called nodal lines and nodal planes. Often we may need to introduce additional node points along the nodal lines or planes. For structural problems involving static or dynamic applied loads we will be defining the behaviour of the structure in terms of displacements and/or stresses. Within each of our elements we need to select a pattern or shape for the unknown displacement or stress. In the case of a displacement field the shape function defines the behaviour of displacements within an element in terms of unknown quantities specified at the element nodes. These nodal values are known as nodal connection quantities and allow the deformation behaviour in one element to be communicated to adjacent elements. In the case of an assumed stress field in the element the connection quantities are different but the underlying principle is the same.

At once we see some of the power of the finite element method because, for a specific element type (beam, plate, shell, etc.), the shape functions are identical for each element. Thus, a given element need only be programmed once and the computer can repeat the operations specified for one, general, element as often as required.

The structure is clearly going to be modelled by an assemblage of finite elements but this introduces a number of problems which need resolving. How does one actually numerically define a finite element in terms suitable for the computer? How do we select elements which will be accurate enough to adequately represent the structural behaviour? How does one apply the design loads in a finite element analysis? How do we select the correct connection quantities? How do we tell the computer to assemble a collection of individual elements so that the actual structural behaviour represented is modelled? The answer to these and other problems represent the 'Rules of the Game' for the finite element method. The engineer conversant with these 'Rules' is equipped to use the finite element method for the solution of real industrial based design problems. Furthermore, he is able to use effectively the major structural analysis finite element programs such as NASTRAN, PAFEC , FINEL ASAS, which have become a routine part of structural analysis in all major industries. These 'Rules' and their explanation form the basis of the current course.

1.1 Element types

Thus we have seen that the essence of the finite element method is to use a piecewise continuous approximating function. As already indicated the selection of the terms to be approximated dictates the connection quantities:

(i) Displacement Elements - these are usual elements found in the major system commonly employed to perform structural analyses. In this case the displacement field at all points on the interior and the boundary of the element is approximated in terms of a low order polynomial. As we shall see this gives rise to connection quantities in terms of displacements and their derivatives defined at nodes on the element boundary.

(ii) Equilibrium Elements - for this element the stresses, usually defined in terms of stress functions, are approximated by low order polynomials. The result of this operation is to give rise to side connection quantities. These can be awkward to handle and it is sometimes possible to produce displacement type connection quantities. Providing the procedure for creating such connections is properly followed the resulting elements are still pure equilibrium elements.

(iii) Hybrid Elements - these elements use two separate approximating fields to describe the elements behaviour usually employing one approximating scheme for the interior of the element with a second being employed on the boundary. For example, the stresses on the interior might be approximated by one set of polynomials with a line approximation being made to displacements along the element boundary. This second displacement field is, in fact, playing the role of a Lagrange multiplier which is attempting to preserve continuity of stress across the element boundaries. In this case the connectors are displacements and/or their derivatives at specific nodes on the element boundary.

1.2 The Direct Approach

The earliest development of the finite element method involved the use of matrix methods on, essentially, simple structural forms. This process required that the main matrices and vectors were built-up stage by stage using elasticity theory as the basic template. The method does not require variational and other principles which latter became a feature of the finite element method and, thus, it became known as the Direct Approach.

We shall use the Direct Approach to give an overview of the main features of the finite element method in an illustrative rather than rigorous manner.

2. THE ELEMENT STIFFNESS MATRIX

To begin, take a simple spring, as shown in Fig.2.1, where the displacements at the two end nodes are given by Δ_1, Δ_2 and the corresponding forces by P_1 and P_2.

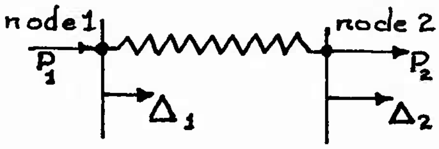


Figure 2.1

Hookes law for this structure gives

$$P_1 = k(\Delta_1 - \Delta_2)$$

$$P_2 = k(\Delta_2 - \Delta_1)$$

where k is the stiffness of the spring.

Using matrix notation

$$\tilde{P} = \tilde{k} \cdot \tilde{\Delta}$$

The stiffness matrix $\tilde{k}$ has two important properties which are clearly demonstrated, that is, it is both Symmetric and Singular.

In order to start the process of generalisation we write $\tilde{k}$ in the form

$$K = \begin{Bmatrix} k_{11} & k_{12} \\ k_{12} & k_{22} \end{Bmatrix}$$

Thus k_{11} represents a stiffness coefficient equal to the force required at node 1 to produce a unit deflection at node 1. Similarly, k_{12} is the force required at node 1 (or 2) to produce a unit deflection at node 2 (or 1).

Our next example involves the linear axial shown at Fig.2.2. The principle for developing the stiffness matrix is identical to that of the above linear spring where

$$k = \frac{AE}{L}$$

Hence $$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \frac{AE}{L} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix} \qquad (2.1)$$

2.1 Co-ordinate Transformation

So far we have tacidly assumed that the co-ordinates of the bar and the co-ordinate system in which the bar is embedded are the same. This situation will not normally prevail and we must consider the bar situated in a non-coincidental co-ordinate system. For convenience we shall take a two-dimensional Cartesian co-ordinate system and consider the bar situated as shown in Fig.3. The bar now has two forces and displacements associated with each of the two end nodes. The associated stiffness matrix must be a 4x4 matrix as opposed to the 2x2 matrix. Thus the relationship between the forces F_i and the displacements δ_i are given via the 4x4 stiffness thus

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ & k_{22} & k_{23} & k_{24} \\ & & k_{33} & k_{34} \\ & & & k_{44} \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix}$$

In order to evaluate the terms k_{ij} we recall that these represent the forces associated with unit nodal displacements. Thus to find k_{11} we set $\delta_1=1$ and $\delta_2=\delta_3=\delta_4=0$, as shown in Fig.2.4. Then the strain in the bar is given by $\delta_1\cos\Theta/L$ which for $\delta_1=1$ becomes $\cos\Theta/L$. Hookes law now gives the force in the bar

$$P = \frac{EA\cos\Theta}{L}$$

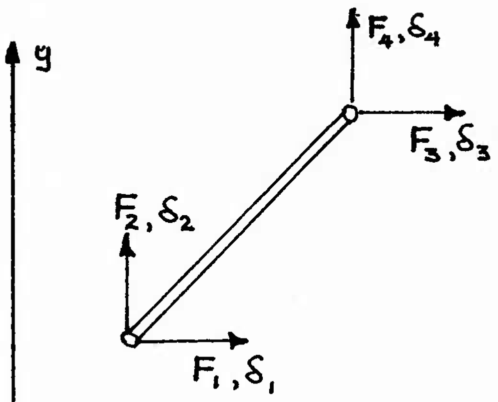


Figure 2.3

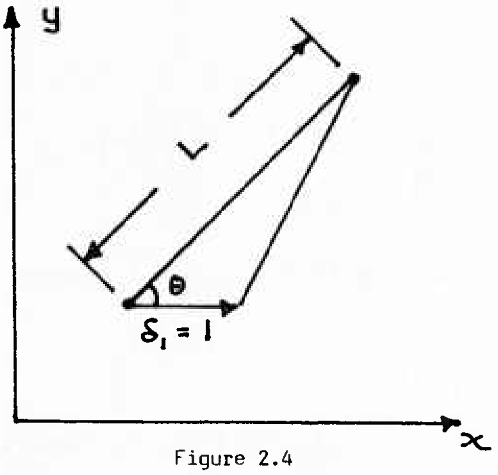


Figure 2.4

We can now resolve P into components F_i sic:

$$F_1 = P\cos\Theta = \frac{EA\cos^2\Theta}{L}$$

$$F_2 = P\sin\Theta = \frac{EA\cos\Theta\sin}{L}$$

etc.

Therefore,

$$k_{11} = \frac{EA\cos^2\Theta}{L}$$

$$k_{12} = k_{21} = \frac{EA\cos\Theta\sin\Theta}{L}$$

etc.

Hence

$$\tilde{k} = \frac{AE}{L} \begin{bmatrix} \cos^2\Theta & \cos\Theta\sin\Theta & -\cos^2\Theta & -\cos\Theta\sin\Theta \\ & \sin^2 & -\cos\Theta\sin\Theta & -\sin^2\Theta \\ & & \cos^2 & \cos\Theta\sin\Theta \\ \text{SYM} & & & \sin^2\Theta \end{bmatrix}$$

which is simply the earlier 2x2 transformed into the x-y co-ordinate system. In fact a transformation of this type may be achieved by pre and post multiplying the 2x2 matrix by a transformation matrix. In order to demonstrate this, consider two systems shown in Fig.2.5 where P_i, Δ_i, i=1,2 represent the FE nodal forces and displacements in the local bar system whilst F_i, δ_i, i=1,2,3,4 represent the same terms in the global (x,y) system. Thus we have $\delta_1 = \Delta_1\cos\Theta$, $\delta_2 = \Delta_1\sin\Theta$, $\delta_3 = \Delta_2\cos\Theta$, $\delta_4 = \Delta_2\sin\Theta$.

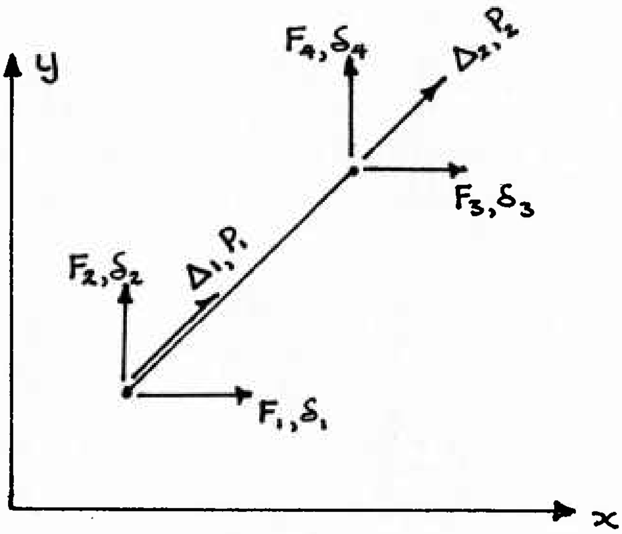


Figure 2.5

Hence

$$\begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix} = \begin{Bmatrix} \cos\Theta & 0 \\ \sin\Theta & 0 \\ 0 & \cos\Theta \\ 0 & \sin\Theta \end{Bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

and

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{Bmatrix} \cos\Theta & 0 \\ \sin\Theta & 0 \\ 0 & \cos\Theta \\ 0 & \sin\Theta \end{Bmatrix} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Now going back to the original element formulation

$$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \frac{AE}{L} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

and pre-multiplying by the transformation matrix

$$\begin{Bmatrix} \cos\Theta & 0 \\ \sin\Theta & 0 \\ 0 & \cos\Theta \\ 0 & \sin\Theta \end{Bmatrix} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \frac{AE}{L} \begin{Bmatrix} \cos\Theta & 0 \\ \sin\Theta & 0 \\ 0 & \cos\Theta \\ 0 & \sin\Theta \end{Bmatrix} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} \Delta_1 \\ \Delta_2 \end{Bmatrix}$$

if we substitute for $\Delta_1 \Delta_2$ using the transformation given above:-

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \frac{AE}{L} \begin{Bmatrix} \cos\Theta & 0 \\ \sin\Theta & 0 \\ 0 & \cos\Theta \\ 0 & \sin\Theta \end{Bmatrix} \begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix} \begin{Bmatrix} \cos\Theta & \sin\Theta & 0 & 0 \\ 0 & 0 & \cos\Theta & \sin\Theta \end{Bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix}$$

$$= \frac{AE}{L} \begin{Bmatrix} \cos^2\Theta & \cos\Theta \ \sin\Theta & -\cos^2 & -\cos\Theta \ \sin\Theta \\ & \sin^2\Theta & -\cos\Theta\sin\Theta & -\sin^2\Theta \\ & & \cos^2\Theta & \cos\Theta \ \sin\Theta \\ & & & \sin^2\Theta \end{Bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix}$$

2.2 Generalised Co-Ordinate Transformation

The previous section illustrated that co-orinate transformation can be achieved in a relatively simple manner. However we would like to reformulate this process in a manner more suitable for computer implementation. For convenience we shall employ the notation shown in Fig.2.6.

Thus taking x,y,v,u as the global system and x^1,y^1,v^1,u^1 as the local system the transformation between the two is given by the equations

$$u^1 = u\cos\alpha + v\sin\alpha$$

$$v^1 = v\sin\alpha + v\cos\alpha$$

Thus

$$\begin{Bmatrix} u^1 \\ v^1 \end{Bmatrix} = \begin{Bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{Bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}$$

or

$$\tilde{d}^1 = \tilde{T}\tilde{d}$$

where $\tilde{T}$ is the transformation matrix

$$\begin{Bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{Bmatrix}$$

If we now consider a bar element with nodes (1) and (2) as shown in Fig.2.6 then

$$\tilde{d}^1 = \begin{Bmatrix} u_1{}^1 \\ v_1{}^1 \\ u_2{}^1 \\ v_2{}^1 \end{Bmatrix} \text{ and } \tilde{d} = \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix}$$

and $\tilde{d}^1 = \tilde{T}\tilde{d}$

where

$$T = \begin{Bmatrix} \tilde{\Lambda} & \tilde{0} \\ \tilde{0} & \tilde{\Lambda} \end{Bmatrix}$$

with

$$\tilde{\Lambda} = \begin{Bmatrix} \cos & \sin\alpha \\ -\sin & \cos\alpha \end{Bmatrix}$$

For three dimensions the matrix $\tilde{\Lambda}$ becomes a 3x3 matrix and other terms will appear if we have bending as opposed to tension elements.

The matrix $\tilde{T}$ is orthogonal and thus

$$\tilde{T}^{-1} = \tilde{T}^t$$

therefore

$$\tilde{d} = \tilde{T}^t\tilde{d}^1$$

Thus the stiffness matrix can be transformed from local to global co-ordinates, to perform this transofrmation we start with (2.1) written in global co-ordinates and in the notation of this section:-

$$\tilde{k}\tilde{d} = \tilde{p}$$

In order to convert the load and nodal variables we use the above transformation matrix thus

$$\tilde{d} = \tilde{T}^t\tilde{d}^1 \text{ and } \tilde{p}^1 = \tilde{T}\tilde{p}$$

thus $\tilde{T}\tilde{k}\tilde{d} = \tilde{T}\tilde{p} = \tilde{p}^1$

and then

$$\tilde{T}\tilde{k}\tilde{T}^t\tilde{d}^1 = \tilde{p}^1$$

thus $\tilde{T}\tilde{k}\tilde{T}^t = \tilde{k}^1$

or $\tilde{k} = \tilde{T}^{-1}\tilde{k}^1(\tilde{T}^t)^{-1} = \tilde{T}^t\tilde{k}^1T$

and transformation for the element stiffness matrix from local to global co-ordinates is given by

$$\tilde{k} = \tilde{T}^t\tilde{k}^1\tilde{T}$$

2.3 Assembling Elements

Now that we can transform an element from a local to a global co-ordinate system we can think in terms of assembling a group of elements to represent the behaviour of a structure. Consider the spring assemblage shown in Fig.2.7 consisting of 6 springs and 6 nodes with 12 degrees of freedom. In order to see how the 6 individual elements are assembled into a global representation we need only consider what happends when we take 2 elements with a common mode. To this purpose we take elements (1) and (2) which share a common node at 4.

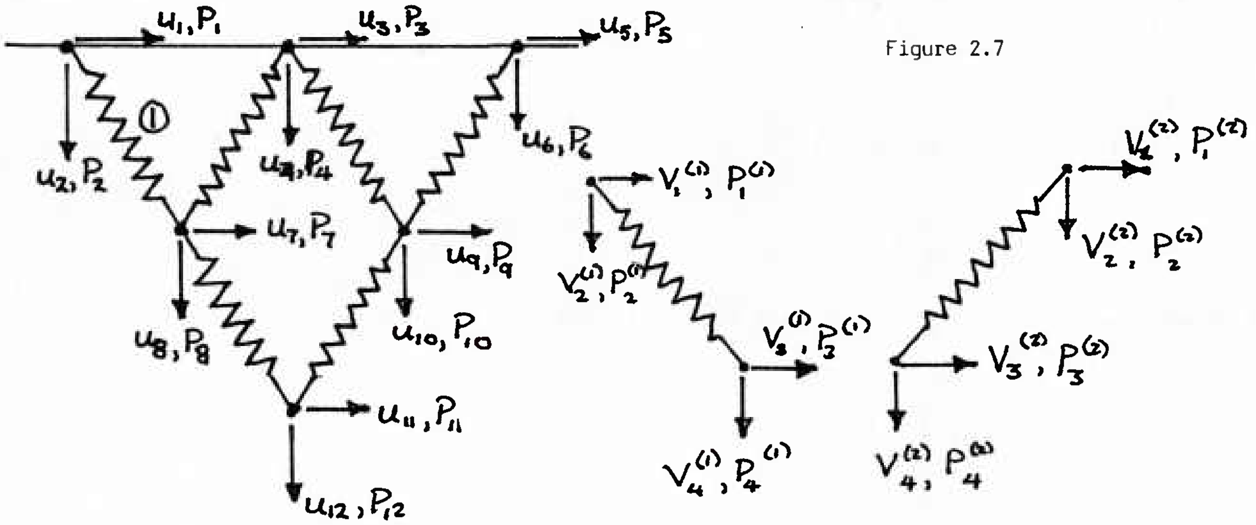


Figure 2.7

Assembling

$$
\begin{array}{cccccccccccc}
1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 & 9 & 10 & 11 & 12
\end{array}
$$

$$
\left\{\begin{array}{cccccccccccc}
k_{11}^{(1)} & k_{12}^{(1)} & & & & & k_{13}^{(1)} & k_{14}^{(1)} & & & & \\
k_{21}^{(1)} & k_{22}^{(1)} & & & & & k_{23}^{(1)} & k_{24}^{(1)} & & & & \\
 & & k_{11}^{(2)} & k_{12}^{(2)} & & & k_{13}^{(2)} & k_{14}^{(2)} & & & & \\
 & & k_{21}^{(2)} & k_{22}^{(2)} & & & k_{23}^{(2)} & k_{24}^{(2)} & & & & \\
 & & & & & & & & & & & \\
 & & & & & & & & & & & \\
k_{31}^{(1)} & k_{32}^{(1)} & k_{31}^{(2)} & k_{32}^{(2)} & & & k_{33}^{(1)}+k_{33}^{(2)} & k_{34}^{(1)}+k_{34}^{(2)} & & & & \\
k_{41}^{(1)} & k_{42}^{(1)} & k_{41}^{(2)} & k_{42}^{(2)} & & & k_{34}^{(1)}+k_{43}^{(2)} & k_{44}^{(1)}+k_{44}^{(2)} & & & & \\
 & & & & & & & & & & & \\
 & & & & & & & & & & & \\
 & & & & & & & & & & & \\
 & & & & & & & & & & &
\end{array}\right\}
\left\{\begin{array}{c} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \\ u_7 \\ u_8 \\ u_9 \\ u_{10} \\ u_{11} \\ u_{12} \end{array}\right\}
=
\left\{\begin{array}{c} P_1^{(1)} \\ P_2^{(1)} \\ P_1^{(2)} \\ P_2^{(2)} \\ \\ \\ P_3^{(1)}+P_3^{(2)} \\ P_4^{(1)}+P_4^{(2)} \\ \\ \\ \\ \\ \end{array}\right\}
=
\left\{\begin{array}{c} P_1 \\ P_2 \\ P_3 \\ P_4 \\ P_5 \\ P_6 \\ P_7 \\ P_8 \\ P_9 \\ P_{10} \\ P_{11} \\ P_{12} \end{array}\right\}
$$

FIGURE 2.8

DES 8422/8

In Fig.2.7 we have shown the two elements separated out from the total assemblage with their own independent load and displacement system $P_1^{(i)}$, $P_2^{(i)}$, $P_3^{(i)}$, $P_4^{(i)}$ etc. with i=1 or 2 dependent upon the element.

In assembling the global stiffness matrix we must add into the 12x12 matrix, which relates the load $P_1...P_{12}$ to the displacements $u_1...u_{12}$, the individual 4x4 stiffness matrices for each of the elements such as (1) and (2). Thus we look at node 4 and observe that

$$P_7 = P_3^{(1)} + P_3^{(2)}$$

$$P_8 = P_4^{(1)} + P_4^{(2)}$$

which requires that the appropriate contributions from the individual element stiffness matrices are added

$$P_7 = k_{31}^{(1)}V_1^{(1)} + k_{32}^{(1)}V_2^{(1)} + k_{33}^{(1)}V_3^{(1)} + k_{34}^{(1)}V_4^{(1)}$$
$$+ k_{31}^{(2)}V_1^{(2)} + k_{32}^{(2)}V_2^{(2)} + k_{33}^{(2)}V_3^{(2)} + k_{34}^{(2)}V_4^{(2)}$$

or in global terms, since $V_1^{(2)} = U_1$, $V_2^{(1)} = U_2$, $V_3^{(1)} = V_3^{(2)} = U_7$; $V_4^{(1)} = V_4^{(2)} = U_8$

$$P_7 = k_{31}^{(1)}U_1 + k_{32}^{(1)}U_2 + (k_{33}^{(1)} + k_{33}^{(2)})\ U_7 + (k_{34}^{(1)} + k_{34}^{(2)})\ U_8$$
$$+ k_{31}^{(2)}U_3 + k_{32}^{(2)}U_4$$

and similarly for P_8. When these are transcribed into the global stiffness matrix we obtain the form shown in Fig.2.8. By proceeding in an identical manner for each of the nodes, the full global stiffness matrix and load vector are assembled. In reality the load vector contains the actual loads on the structure and is not assembled in the manner shown.

2.4 Incorporation of Boundary Conditions

Having assembled the global stiffness matrix and load vector we are still not in a position to solve for the unknown global displacements. We must now apply the boundary conditions which form part of the problem definition and which also fix the structure in space and prevent rigid body rotation. It is worth noting that if all the rigid body modes are not fixed then the element and global stiffness matrices are singular. Once the boundary conditions are prescribed the structural analysis problem can be solved by inversion of the global stiffness matrix or by employing some other equation solving procedure.

As we have seen throughout our developments of the F-E method because we are dealing with a displacement formulation, we can only specify boundary conditions on the boundary S_u. In the F-E method these conditions are defined in terms of specified nodal displacements which may be zero if the structure is attached to a rigid support or may be non-zero if some movememnt of the supports or attachment points is specified.

In order to demonstrate the procedure for specifying boundary displacements and the subsequent procedures for solving we divide the global displacement vector by partitioning into free nodal displacements $\tilde{U}$ and specified displacements $\tilde{U}^*$. The global stiffness matrix and load vector must also be similarly partitioned thus

$$\begin{Bmatrix} \tilde{K}_{11} & \tilde{K}_{12} \\ \tilde{K}_{21} & \tilde{K}_{22} \end{Bmatrix} \begin{Bmatrix} \tilde{U} \\ \tilde{U}^* \end{Bmatrix} = \begin{Bmatrix} \tilde{P} \\ \tilde{P}^* \end{Bmatrix}$$

where the vector $\tilde{P}^*$ represent the reactions at the nodes where the displacements $\tilde{U}^*$ are specified.

Expanding

$$\tilde{K}_{11}\tilde{U} + \tilde{K}_{12}\tilde{U}^* = \tilde{P}$$
$$\tilde{K}_{21}\tilde{U} + \tilde{K}_{22}\tilde{U}^* = \tilde{P}^*$$

Since the displacements $\tilde{U}$ are the only unknowns then we can solve for these to give:

$$\tilde{U} = \tilde{K}_{11}^{-1}\left\{\tilde{P} - \tilde{K}_{12}\tilde{U}^*\right\} \qquad (2.2)$$

and the reactions are given by:

$$\tilde{P}^* = \tilde{K}_{21}\tilde{K}_{11}^{-1}\tilde{P} + \left\{\tilde{K}_{22} - \tilde{K}_{21}\tilde{K}_{12}\tilde{K}_{12}\right\}\tilde{U}^* \qquad (2.3)$$

If $U^* = 0$ then we have

$$U = \tilde{K}_{11}^{-1}\tilde{P}$$

$$\tilde{P}^* = \tilde{K}_{21}\tilde{K}_{11}^{-1}\tilde{P}$$

Although for convenience we have assumed that the specified terms in the displacement vector are situated at the bottom of the vector column. In reality, this situation will never arise and could only be achieved by re-numbering the structure. However, for the case when $\tilde{U}^* = 0$ the scatering of the components of $\tilde{U}^*$ within the displacement vector, causes no computational problem. All we do is simply remove each row and column associated with each component $\tilde{U}_i^* = 0$. Thus for each i we delete the ith row and column from the global stiffness matrix $\tilde{K}$ but preserve in store the terms for the matrix $\tilde{K}_{12}$ to allow for subsequent recovery of the reactions. The way in which this is handled for our spring problem is shown in Fig.2.9.

An equivalent technique is available for the case when $\tilde{U}^* \neq 0$ in this case zero's are placed in the appropriate rows and columns of $\tilde{K}$ and a 1 placed on the diagonal. Thus:-

if U_{ji} is prescribed as V^*_{ji}

then $K_{ji} = K_{ij} = 0$ for $i \neq j$ and $i = 1,2,...n$

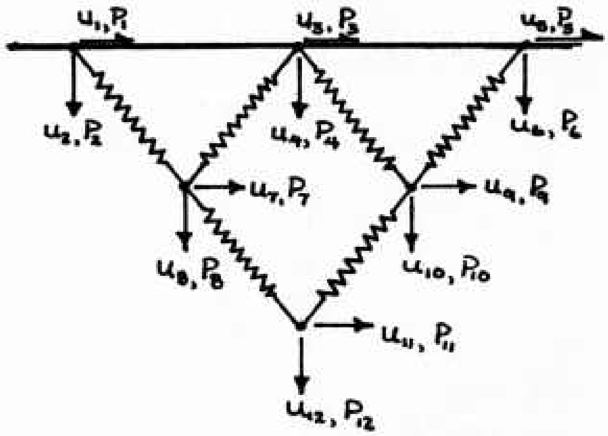
and $J_{ij} = 1$

This is balanced by replacing P_i in the load vector by $P_i - K_{ij}U^*_i$ for $i = 1,...n$. This operation is equivalent to a partitioned matrix

$$\begin{Bmatrix} \tilde{K}_{11} & \tilde{0} \\ \tilde{0} & I \end{Bmatrix} \begin{Bmatrix} \tilde{U} \\ \tilde{U}^* \end{Bmatrix} = \begin{Bmatrix} \tilde{P} - \tilde{K}_{12}\tilde{U}^* \\ \tilde{U}^* \end{Bmatrix}$$

In order to remove the singularity property for the simple problem, we must prescribe some of the displacements $U_1 ... U_{12}$ and for the present example the displacements U_1 through U_6 are zero because the structure is rigidly held at these points as shown in Fig.2.9. Because these are zero conditions we can employ 2.3 and delete the row and column associated with each prescribed displacement. This deflated matrix is no longer singular and the problem can now be solved to yield values for the unknown displacements U_7 to U_{12} and values for the unknown reactions P_1 to P_6.

Boundary Conditions



$$\begin{Bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} & K_{17} & K_{18} & K_{19} & K_{110} & K_{111} & K_{112} \\ & K_{22} & & & & & & & & & & \\ & & K_{33} & & & & & & & & & \\ & & & K_{44} & & & & & & & & \\ & & & & K_{55} & & & & & & & \\ & & & & & K_{66} & & & & & & \\ & & & & & & K_{77} & & & & & \\ & & & & & & & K_{88} & & & & \\ & & & & & & & & K_{99} & & & \\ & \text{SYM} & & & & & & & & K_{110} & & \\ & & & & & & & & & & K_{111} & \\ & & & & & & & & & & & K_{112} \end{Bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ U_{12} \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ '' \\ P_{12} \end{Bmatrix}$$

Figure 2.9

3. THE DISPLACEMENT FINITE ELEMENT FORMULATION

Although the direct method shows all the main steps which a finite element system passes through in achieving a solution to a structural analysis problem certain very important aspects of element development cannot be covered by this approach. For example, we have no way of deciding what connection quantities are appropriate to a specific element. Nor can we compute the load vectors for the element when subjected to external forces. The direct approach does not provide any information concerning the generation of more general element stiffness matrices.

The way around these difficulties is to employ one of the oldest techniques available to the applied mechanics specialist - the variational method. Unfortunately, time does not allow us to explore this interesting method and show its full power. This may be fortunate since the full intricacies of the method do require careful study and can be difficult to understand. However, if we concentrate on developing and applying displacement elements only we can exploit this method to achieve the above stated requirements without too much strain.

For displacement elements the variational formulation devolves down to requiring the potential energy functional which can then be differentiated to yield the solution state. Because we know that the solution state for an elastic body is terms of the displacement field is that which minimises the potential energy term.

The potential energy π_p is defined as the sum of the internal strain energy U and the potential of the external loads. However this latter term is more conveniently considered as the opposite of the work done by the loads W thus,

$$\pi_p = U - W$$

In the case of the simple bar shown in Fig.3.1 the external work is given by Pu and the strain energy, as usual, by $\frac{1}{2}E\varepsilon^2 a\ell$ with E Youngs Modulus and ε the strain $= \frac{u}{\ell}$.

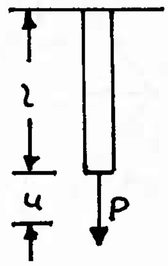


Figure 3.1

Hence

$$\pi_p = U - W = \frac{1}{2}E\varepsilon^2 a\ell - Pu.$$

If we now minimise this term for the only free variable which we have, namely the tip displacement u, then:

$$\frac{\partial \pi_p}{\partial u} = E\varepsilon a\ell \frac{d\varepsilon}{du} - P$$

now $\quad \varepsilon = \frac{u}{\ell} \therefore \frac{d\varepsilon}{du} = \frac{1}{\ell}$

thus $\quad P = E\varepsilon a \quad$ the simple equilibrium solution.

3.1 Shape Functions

In constructing an element we don't wish to be concerned with functions along element boundaries and even less do we want to be concerned with what goes on in the interior of an element. What we want is to be able to describe the displacement field in terms of discrete quantities at specific points in the element called nodes. These nodes are normally located on the element boundaries and connect one element to another.

If we wish to use only nodal values for displacements then we need functions which describe the displacements at all points within the element and on its boundary in terms of these nodal values. The resulting functions are called <u>shape functions</u>. As an example of a shape function we consider the beam element where the connections are taken to be the normal displacements and the rotations as shown in Fig.3.2. Taking a cubic interpolation function then

$$w = a_1+a_2x+a_3x^2+a_4x^3$$

setting

$$w = w_1 \quad \text{at } x = 0$$
$$\theta = \theta_1$$
$$w = w_2 \quad \text{at } x = L$$
$$\theta = \theta_2$$

We can solve for the a_i

$i = 1,2,3,4$, in terms of $w_1, w_2, \theta_1, \theta_2$:

$$\tilde{d} = \tilde{A}\tilde{a} \quad \text{where } \tilde{d}^T = \{w_1,\theta_1,w_2,\theta_2\}^T$$
$$\tilde{a}^T = \{a_1,a_2,a_3,a_4\}^T$$

and thus the displacement w can be defined at all points on the beam through the expression

$$w = \tilde{N}\tilde{d}$$

where $\tilde{N}$ is given by $\{N_1N_2N_3N_4\}$ where

$$N_1 = 1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}, \qquad N_3 = \frac{3x^2}{L^2} - \frac{2x^3}{L^3},$$

$$N_2 = x - 2x^2/_L + x^3/_{L^2}, \qquad N_4 = -\frac{x^2}{L} + \frac{x^3}{L^3}$$

Thus,

$$w = \left(1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}\right)w_1 + \left(x - \frac{2x^2}{L} + \frac{x^3}{L^2}\right)_1 + \left(\frac{3x^2}{L^2} - \frac{2x^3}{L^3}\right)$$

$$+ \left(-\frac{x^2}{L}\ \frac{x^2}{L^2}\right)\theta_2$$

The elements of the vector $\tilde{N}$, i.e. N_i i=1,2,3,4 are shape functions relating the displacement w at all points within the element in terms of the nodal connection qualities $w_1, \theta_1, \theta_2 \theta_2$.

The functions N_1, N_3 take unit values at one end of the beam .nd zero at the other and are, in fact, Hermitian interpolation functions. Thus if we had not wished to go through the derivation process we could have simply noted that we require functions taking unit value at a specified point reducing to zero at the other end of the interval and turned immediately to Hermitian functions. For co-continuity one can use Lagrangian interpolation.

3.2 Finite Element Formulation

In describing the concept of shape functions we have employed displacement functions only and this is maintained in the current section where element stiffness matrices are derived. Thus we only look at the displacement element formulation and leave element types such as equilibrium and hybrid formulation to other texts. This simplifying assumption is made for two reasons. First, the displacement element formulation is by far the most popular approach and is employed by all the major finite element analysis system. Second, the principles which underlie the development of displacement finite element stiffness matrices apply to these other types even though the basic functional employed may be different.

In order to develop the stiffness matrix for a given displacement finite element we assume that we have an approximate form for the displacement field within the element given by $\tilde{u}$. We also assume that we may have body forces $\tilde{F}$ acting within the element and that surface tractions $\tilde{T}$ may be applied over a region of the element surface defined by S_σ. Under all these assumptions the potential energy term for an isolated element is obtained by generalising the potential energy term π_p introduced in the first section to yield:

$$\pi_p = \iiint \frac{1}{2}\tilde{\varepsilon}^T\tilde{D}\tilde{\varepsilon}dv - \iiint \tilde{F}^T\tilde{u}dv - \iint \tilde{T}^T\tilde{u}ds$$

where D is the strain-strain matrix standing in place of E and the strain energy density term for the element is given by

$$\frac{1}{2}\tilde{\varepsilon}^T\tilde{D}\tilde{\varepsilon}$$

we may define the strain-displacement relationships by the matrix expression

$$\tilde{\varepsilon} = \tilde{B}\tilde{u}$$

then using the shape function $\tilde{\varepsilon} = \tilde{B}\tilde{N}\tilde{d}$

the strain energy becomes

$$\frac{1}{2}\tilde{\varepsilon}^T\tilde{D}\tilde{\varepsilon} = \frac{1}{2}\tilde{d}^T(\tilde{B}\tilde{N})^T\tilde{D}(\tilde{B}\tilde{N})\tilde{d}$$

and we define $\iiint(\tilde{B}\tilde{N})^T\tilde{D}(\tilde{B}\tilde{N})\tilde{d}\tilde{v}$ as the <u>element stiffness matrix</u>

$$\iiint\frac{1}{2}\tilde{\varepsilon}^T\tilde{D}\tilde{\varepsilon}dv = \frac{1}{2}\tilde{d}^T\tilde{k}\tilde{d} \qquad (3.1)$$

Turning next to the forces on the element which represents a generalisation of the work term in Pu for the bar and using the shape function matrix we have

$$\iiint\tilde{F}^T\tilde{u}dv + \iint_{s\sigma}\tilde{T}^T\tilde{u}ds$$

$$= \iiint\tilde{d}^T\tilde{N}^T\tilde{F}dv + \iint_{s\sigma}\tilde{d}^T\tilde{N}^T\tilde{T}ds$$

$$= \tilde{d}^T\tilde{p}$$

where $\tilde{p} = \iiint \tilde{N}^T\tilde{F}dv + \iint_\sigma \tilde{N}^T\tilde{T}ds$ (3.2)

Thus the potential energy for an isolated element is given by the expression

$$\pi_p = \frac{1}{2}\tilde{d}^T\tilde{k}\tilde{d} - \tilde{d}^T\tilde{p}$$

In order to evaluate the unkown nodal displacements $\tilde{d}$ we apply the variation $\delta\pi_p = 0$ which, in this case, requires that we differentiate π_p in terms of the displacements Hence

$$\frac{\partial\pi_p}{\partial\tilde{d}} = \tilde{k}\tilde{d} - \tilde{p} = 0$$

or $\tilde{k}\tilde{d} = \tilde{p}$

Having developed the basis of our theory for creating the finite element stiffness matrix $\tilde{k}$ and the consistent load vector $\tilde{p}$ its appropriate to look at some simple examples.

3.2.1 Examples

1. Bar Element (axial)

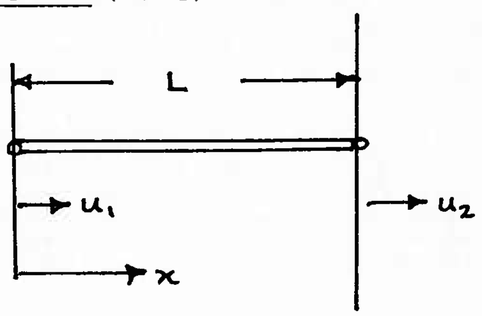


Figure 3.2

Consider the first two nodal axial or bar element of length L and cross sectional area A, which we used in earlier sections. At node (1) we have a nodal displacement u_1 and u_2 is at node (2).

We want a shape function which allows the displacement to take a value of unity at node (1) reducing to zero at node (2) and a second function which takes a value of unity at (2) reducing to zero at (1). These functions are shown in Fig.3.2 but could have been found by demanding a linear variation in u along the bar, i.e.

$$u = a_1 + a_2 x$$

Thus $N = \left\{\frac{L-x}{L} \;\; \frac{x}{L}\right\}$

For a bar $\epsilon = {}^{\partial u}/_{\partial x}$ $\therefore$ $B = {}^{\partial}/_{\partial x}$

$\therefore$ $\tilde{B}\tilde{N} = \left\{\frac{-1}{L} \;\; \frac{1}{L}\right\}$

and $\tilde{D} = E$

$$(\tilde{B}\tilde{N})^T\tilde{D}(\tilde{B}\tilde{N}) = \begin{Bmatrix} \frac{-1}{L} \\ \frac{1}{L} \end{Bmatrix} \left\{\frac{-E}{L} \;\; \frac{E}{L}\right\}$$

$$= \frac{E}{L^2}\begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}$$

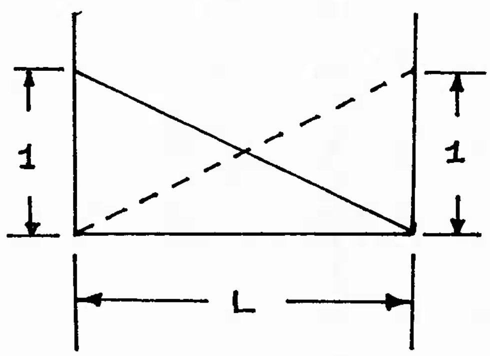


Figure 3.3

$$\tilde{k} = \int_0^L (\tilde{B}\tilde{N})^T \tilde{D} (\tilde{B}\tilde{N}) A dx = \frac{AE}{L}\begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}$$

As we had before!

In order to demonstrate how to calculate $\tilde{p}$ let us assume that a concentrated force (c) of 1 units is applied at $x = {}^{2L}/_3$ in axial directions, a friction force caused by air flowing along the bar produces a unit force/unit length (q).

We have no body forces $\tilde{F}$ only traction forces $\tilde{T}$ applied over the surface as in the case of the traction forces and at a point as in the case of the concentrated load thus,

$$P = \int_0^L \tilde{N}^T q dx + \tilde{N}^T c$$

as both c and q take unit values;

$$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \int_0^L \begin{Bmatrix} \frac{L-x}{L} \\ \frac{x}{L} \end{Bmatrix} dx + \begin{Bmatrix} {}^1/_3 \\ {}^2/_3 \end{Bmatrix}$$

$$\begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix} = \begin{Bmatrix} \frac{L}{2} \\ \frac{L}{2} \end{Bmatrix} + \begin{Bmatrix} {}^1/_3 \\ {}^2/_3 \end{Bmatrix}$$

Thus the load applied at node (1) is $P_1 = {}^1/_3$ and at node (2) $P_2 = \frac{L}{2} + {}^2/_3$. This technique for generating nodal forces gives rise to a consistent set and for the bar these F-E consistent loads are compared with the actual loads in Fig.3.4. Note that the only loads on the element are P_1 and P_2 representing forces at the nodes in the direction of the displacement u_1 and u_2.

2. Beam Element (Bending)

Now we turn to the beam bending problem in which we have nodes with two connection quantities, the nodal displacement w and the rotation θ. From the section 3.1 the shape function matrix is given by

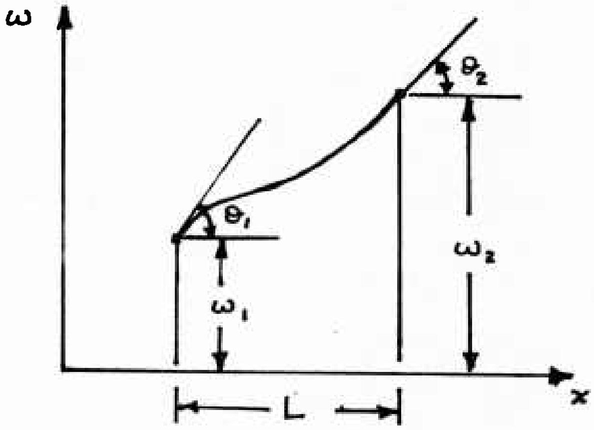


Figure 3.5

$$N = \left\{ \left(1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}\right) \left(x - \frac{2x^3}{L} + \frac{x^3}{L^2}\right) \left(\frac{3x^2}{L^2} - \frac{2x^3}{L^3}\right) \left(-\frac{x^2}{L} + \frac{x^3}{L^2}\right) \right\}$$

For a beam the strain is given by the curvature change

$$k = \partial^2 w/\partial x^2 \qquad \text{i.e.} \quad B = \partial^2/\partial x^2$$

$$\therefore \quad BN = \left\{ \left(\frac{-6}{L^2} + \frac{12x}{L^3}\right) \left(-\frac{4}{L} + \frac{6x}{L^2}\right) \left(\frac{6}{L^2} - \frac{12x}{L^3}\right) \left(-\frac{2}{L} + \frac{6x}{L^2}\right) \right\}$$

For a beam $\tilde{D} = EI$

Thus the strain energy term for the beam is given by

$$\int_0^L \frac{1}{2} EI \left\{\frac{d^2w}{dx}\right\} dx$$

$$\therefore \quad \tilde{k} = \int_0^L (\tilde{B}\tilde{N})^T EI(\tilde{B}\tilde{N})dx = \frac{EI}{L^3}\begin{Bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{Bmatrix}$$

As before we now wish to calculated the nodal forces on the element appropriate to the nodal connection quantities, i.e. we wish to compute

$$\tilde{P} = \iiint \tilde{N}^T\tilde{F}dv + \iint_{s\sigma} \tilde{N}^T\tilde{T}ds.$$

Assuming a self weight force q/unit volume, a point P load applied at the mid-point of the beam and a moment M applied at $x = {}^{2L}/3$ then we have (see Fig.3.6).

$$\tilde{P} = \iiint -\tilde{N}^T q dv + \tilde{N}^T\Big|_{x=L/2} P + \frac{d\tilde{N}^T}{dx}\Big|_{x=2L/3} M$$

we have $d\tilde{N}^T/dx$ because the work term for the moment = MΘ = Mdw/dx

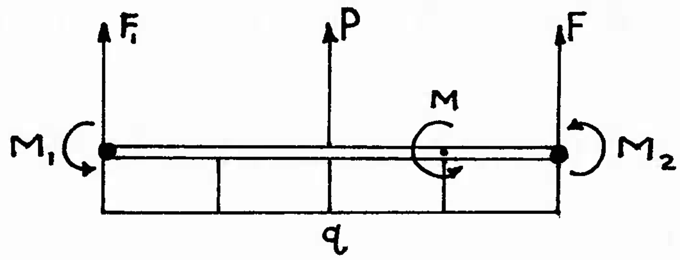


Figure 3.6

If the beam is of rectangular section of width 'A' and depth 'B' then

$$\tilde{P} = \int_0^L -\tilde{N}^T qABdx + \tilde{N}^T\Big|_{x=L/2} P + \frac{dN^T}{dx}\Big|_{x=2L/3} M$$

$$\therefore \quad \tilde{P} = \begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix} = -qAB\begin{Bmatrix} L/2 \\ L^2/12 \\ L/2 \\ -L^2/12 \end{Bmatrix} + P\begin{Bmatrix} 1/2 \\ L/8 \\ L/2 \\ -1/8 \end{Bmatrix} + M\begin{Bmatrix} -\frac{4}{3L} \\ -1/3 \\ \frac{4}{3L} \\ 0 \end{Bmatrix}$$

3.3 Stress Computation

The underlying concept of stress evaluation relies on combining the stress-strain expression

$$\tilde{\sigma} = \tilde{D}\tilde{\varepsilon}$$

with the strain-displacement expression

$$\tilde{\varepsilon} = \tilde{B}\tilde{u} = \tilde{B}\tilde{N}\tilde{d}$$

to give

$$\tilde{\sigma} = \tilde{B}\tilde{D}\tilde{u} = \tilde{B}\tilde{N}\tilde{D}\tilde{d}$$

Of course, this is all done at the element level so that we compute the stresses in each element individually. Using the displacements obtained from the solution process of earlier section, i.e. from,

$$\tilde{V} = \tilde{K}^{-1}\tilde{P}$$

For the simple bar or constant strain triangular elements the strain field, and therefore, the stress field is constant throughout the element. In the case of more complex elements the stress will vary within the element as is the case with the beam element, some decision is required with respect to where we compute the stress. At the element level one may select nodal values for the co-ordinates and obtain nodal stresses. Or one may select an interior point and, thereby, compute element stresses away from the nodes.

There is also the question of how one treats the variations in stress as one moves from element to element.

In practice a variety of procedures are followed to 'make sense' of the stress output, but there is no substitute for common sense and the application of structural knowledge. The 'rules of thumb' are then:-

1) For Nodal Stresses

   i) If the strain is constant within element - unique values

   ii) If the strain varies with co-ordinates, evaluate at interior point (or points) and interpolate.

2) Stress Averaging

   i) Stresses at given nodes will be different for different elements ∴ average (unless 'stress Jump').

   ii) Alternative is to average stresses over a collection.

   iii) Various 'improvement' schemes using iterative methods are avaiable.

A moderate F-E system would offer these, and other options through a post-processor system.

4. ISOPARAMETRIC ELEMENTS

Having looked at the procedure for generating element stiffness matrices and consistent load vectors we still lack a range of effective elements. In order to expand this limited range we shall now examine the widely used isoparametric element which allows us to generate curved elements effective for membranes, plates, shells and solid structures.

The essence of the element formulation is to use a shape function to define the approximate displacement field and to define the element shape. Thus both the displacement field and the element shape are defined in terms of nodal values. Taking the shape function as N then the approximate displacement field $\tilde{u}$ given by $\tilde{u} = \tilde{N}\tilde{d}$. Consequently the position vector $\tilde{p}$ is given by $\tilde{p} = \tilde{N}\tilde{c}$ where $\tilde{c}$ defines the nodal position co-ordinates. An example of an isoparametric is shown in Fig.4.1 where the position $\tilde{p}$ is defined in terms of the position vectors of the nodes 1, 2, 3, 4.

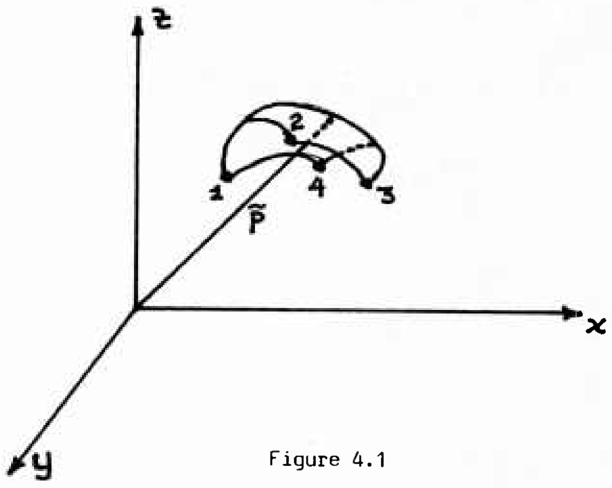


Figure 4.1

4.1 Axial (one-dimensional) Co Bar Elements

It is customery to use natural or intrinsic co-ordinates in developing isoparametric formulations; such co-ordinates range over values ±1. For our one-dimensional problem, illustrated in Fig.4.2, we use $\xi \pm 1$, regardless of the length of the bar. Using the usual linear displacement field for the component u we have

$$u = \tilde{N}\tilde{d} \text{ with } \tilde{N} = \left\{ \frac{1-\xi}{2} \;\; \frac{1+\xi}{2} \right\}$$

$$d = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

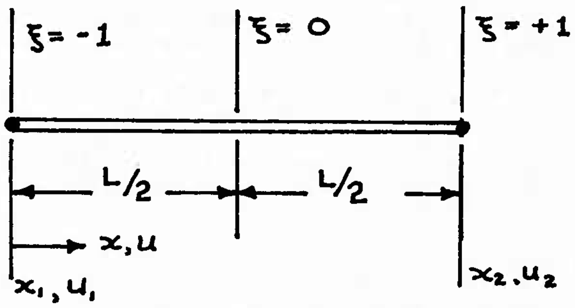


Figure 4.2

Although the co-ordinate ξ ranges of ±1 the element is actually defined in the x-co-ordinate starting at x_1 and continuing to x_2 with a total length of L. Thus a point within the element is given by

$$x = \tilde{N}\tilde{c} \text{ again } \tilde{N} = \left\{ \frac{1-\xi}{2} \;\; \frac{1+\xi}{2} \right\}$$

and $$C = \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix}$$

Thus $$\varepsilon = \frac{2}{L}\frac{du}{d\xi} = \frac{2}{L}\tilde{B}\tilde{N}\tilde{d}$$

The potential energy term for the bar is

$$\pi_p = \int_0^L \frac{1}{2}\tilde{\varepsilon}^T\tilde{D}\tilde{\varepsilon}A\,dx - \int_0^L \tilde{F}^T\tilde{u}A\,dx - \int_0^L \tilde{T}^T\tilde{u}\,dx$$

where A is the bar cross-section area. Substituting for $\tilde{\varepsilon}$ and $\tilde{u}$ gives

$$\pi_p = \int_0^L \frac{1}{2}\tilde{d}^T(\tilde{B}\tilde{N})^T\tilde{D}(\tilde{B}\tilde{N})\tilde{d}A\,dx - \int_0^L \tilde{F}^T\tilde{N}\tilde{d}A\,dx$$

$$- \int_0^L \tilde{T}^T\tilde{N}\tilde{d}\,dx$$

or

$$\pi_p = AE\int_{-1}^{+1} \frac{1}{2}\tilde{d}^T(\tilde{B}\tilde{N})^T(\tilde{B}\tilde{N})\tilde{d}\,J\,d\xi - A\int_{-1}^{+1} \tilde{d}^T\tilde{N}^T F\,J\,d\xi$$

$$- \int_{-1}^{+1} \tilde{d}^T\tilde{N}^T T\,J\,d\xi \qquad (4.1)$$

with, in this case,

$$J = \frac{dx}{d\xi}$$

and $$\frac{dx}{ds} = \frac{dN}{ds} = \frac{x_2 - x_1}{2} = \frac{L}{2}$$

Hence
$$\pi_p = \frac{1}{2}\tilde{d}^T\tilde{k}\tilde{d} - \tilde{d}^T\tilde{p}$$

$$\text{where} \quad K = \int_{-1}^{+1} (\tilde{B}\tilde{N})^T AE(\tilde{B}\tilde{N}) J d\xi = \left(\frac{AE}{L}\right)\begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}$$

$$\tilde{p} = \int_{-1}^{+1} A\tilde{N}^T\tilde{F}Jd\xi + \int \tilde{N}^T\tilde{T}Jd\xi \quad \text{and} \quad J = \frac{L}{2}$$

with $\tilde{F}\xi\tilde{T}$ transformed to the ξ co-ordinate

This two-noded formulation can be generalised to include quadratic (3-noded), cubic (4-noded) as shown in Fig.4.3 or any other higher order element.

quadratic cubic

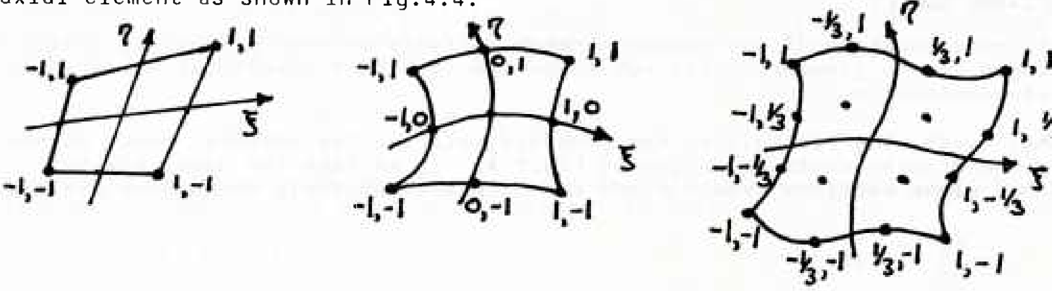
Figure 4.3

The only differences are that the order of the stiffness matrix generated from the potential energy functional 4.1 increases and the form J also changes if the nodes are not equidistant about $\xi = 0$. These higher order shape functions are generated from the Lagrangian interpolation functions given by the formula

$$N_i(x) = \prod_{j=1(j\neq i)}^{n} \frac{\xi - \xi_j}{\xi_i - \xi_j}$$

4.2 <u>Isoparametric Elements in Two and Three-Dimensions with Co-Continuity</u>

The same principles employed above to generate axial elements can be extended to two dimensions without difficulty. In this case we use two natural or intrinsic co-ordinate ξ,η which are used to map the element onto one with unit-length sides. If we consider elements mapped onto the unit square we see the same faily of elements emerge as those for the axial element as shown in Fig.4.4.



Linear Quadratic Cubic

Figure 4.4

As we see the quadratic or higher order elements allow us to model curved boundaries but do generate internal nodes which must be condensed out. These elements use the Lagrangian interpolation function, but multiply together the contributions from each coordinate. For example the shape functions for the linear element case given by

$$N_i(\xi,\eta) = \frac{1}{4}(1+\xi\xi_i)(1+\eta\;_i)$$

thus

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta) \qquad N_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

$$N_3 = \frac{1}{4}(1+\xi)(1+\eta) \qquad N_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

Although Lagrangian functions are a useful way to generate this class of element it is possible to create, directly, elements which do not have interior nodes and these are termed 'serendipity' elements.

In the one dimensional case which we examined above the principle of transforming from one coordinate system to another presented little difficulty. Consider a specific term u which is a function of x and y then the derivatives of this function can be written as

$$\varepsilon_{11} = \frac{1}{\alpha_1}\frac{\partial u_1}{\partial \xi_1} + \frac{1}{\alpha_1\alpha_2}\frac{1}{\partial \xi_2} u_2 + \frac{w}{R_1}$$

$$\varepsilon_{22} = \frac{1}{\alpha_2}\frac{\partial u_2}{\partial \xi_2} + \frac{1}{\partial \xi_2}\frac{\partial \alpha_2}{\partial \xi_1} u_1 + \frac{w}{R_2}$$

$$\varepsilon_{12} = \varepsilon_{21} = \frac{1}{2}\left(\frac{1}{\alpha_1}\frac{\partial u_2}{\partial \xi_1} - \frac{1}{\alpha_1\alpha_2}\frac{\partial \alpha_1}{\partial \xi_2} u_1 + \frac{1}{\alpha_2}\frac{\partial u_1}{\partial \xi_2} - \frac{1}{\alpha_1\alpha_2}\frac{\partial \alpha_2}{\partial \xi_1} u_2\right)$$

$$- \frac{w}{R_{12}}$$

$$k_{11} = \frac{1}{\alpha_1}\frac{\partial \phi_1}{\partial \xi_1} + \frac{1}{\alpha_1\alpha_2}\frac{\partial \alpha_1}{\partial \xi_2}\phi_2$$

$$k_{12} = \frac{1}{\alpha_2}\frac{\partial \phi_2}{\partial \xi_2} + \frac{1}{\alpha_1\alpha_2}\frac{\partial \alpha_2}{\partial \xi_1}\phi_1$$

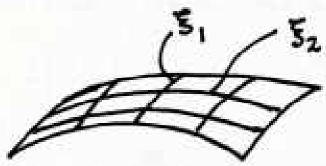


Figure 5.4

$$k_{12} = \frac{1}{\alpha_1}\frac{\partial \phi_2}{\partial \xi_1} - \frac{1}{\alpha_1\alpha_2}\frac{1}{\partial \xi_2}\phi_1 - \frac{\phi_2}{R_1}$$

$$k_{21} = \frac{1}{\alpha_2}\frac{\partial \phi_1}{\partial \xi_2} - \frac{1}{\alpha_1\alpha_2}\frac{\partial \alpha_2}{\partial \xi_1}\phi_2 + \frac{\phi_3}{R_2}$$

and it is common to combine these last two terms to give a symmetric term

$$\bar{k}_{12} = \frac{1}{2}(k_{12} + k_{21})$$

with a_1, α_2 the coefficients of the first fundamental form of the surface. The quantities u_1, u_2 are components of displacement along the co-ordinate directions ξ_1, ξ_2 and w is the displacement along the outwards normal to the shell surface, the terms ϕ_1, ϕ_2, ϕ_3 are the respective rotations as shown in Fig.5.5 R_1, R_2 are the radii of curvature of the median surface with R_{12} the radius of torsion. The strains and curvatures defined above are decoupled but, if the Kirchoff hypothesis is involved a coupling appears because the rotations are then defined in terms of the displacements by

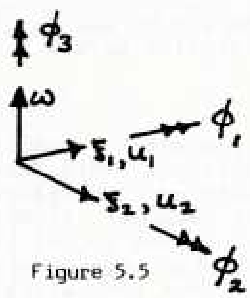


Figure 5.5

$$\phi_1 = \frac{1}{R_1}\left(u_1 - \frac{1}{\alpha_1}\frac{\partial w}{\partial \xi_1}\right)$$

$$\phi_2 = \frac{1}{R_2}\left(u_2 - \frac{1}{\alpha_2}\frac{\partial w}{\partial \xi_2}\right)$$

$$\phi_3 = \frac{1}{2\alpha_1\alpha_2}\left(\alpha_2\frac{\partial u_2}{\partial \xi_1} + u_2\frac{\partial \alpha_2}{\partial \alpha_1} - \alpha_1\frac{\partial u_1}{\partial \xi_2} - u_2\frac{\partial \alpha_1}{\partial \xi_2}\right)$$

The membrane and bending resultants for isotropic materials, associated with this displacement formulation are defined by

$$N_{11} = \frac{12D}{h^2}(\varepsilon_{11} + \nu\varepsilon_{22}) \qquad M_{11} = D(k_{11} + \nu k_{22})$$

$$N_{22} = \frac{12D}{h^2}(\varepsilon_{22} + \nu\varepsilon_{11}) \qquad M_{22} = D(k_{22} + \nu k_{11})$$

$$N_{12} = \frac{12D}{h^2}(1 - \nu)\,\varepsilon_{12} \qquad M_{12} = D(1 - \nu)\,\bar{k}_{12}$$

where h is the shell thickness, D the flexural rigidity

$$D = Eh^3/12(1-\nu^2),$$

E Young's Modulus and ν Poisson's ratio. The boundary conditions from which the consistent load vector may be constructed are, for an arbitrary edge,

$$N_{nn} - \frac{M_{n\nu}}{R_{n\nu}} = N_{nn} \text{ or } U_n = U_n^*$$

$$M_{n\nu} + \frac{M_{n\nu}}{R_\nu} = N^*_{n\nu} \text{ or } U_\nu = U^*_\nu$$

$$M_{nn} = M^{**}_{nn} \; \phi_n \qquad = \phi^*_n$$

$$Q_n + \frac{1}{\alpha_\nu} \frac{\partial M_{n\nu}}{\partial_n} = V^*_n \text{ or } \qquad w = w^*$$

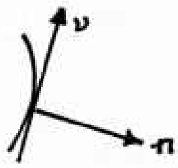


Figure 5.6

Where n, ν are co-ordinates on the median surface which are respectively normal to and along the boundary curve as shown in Fig.5.6; R_n, R_ν, $R_{n\nu}$ are the corresponding radii of curvature.

The central problem of modelling a thin shell element is now exponsed. Unlike the linear theory of flat paltes the equations defining the strain measures, taken in conjunction with the equations specifying the rotations. ϕ_i, i=1,2,3 after the imposition of the Kirchoff, and the form of the boundary equation show that a decoupled bending and stretching stiffness matric is not possible. The problem would be overcome if we could define the couples ϕ_i and the displacements u_1, u_2 and w separately. However, this is not possible if a pure displacement approach is followed and alteratives have been sought. The most popular is that due to the late Brue Irons and known as the semi-loof element. This requires that the corner and mid-side nodes require connection quantitites u_n, u_ν, w and at the loof nodes continuity of ϕ_n is required; as shown in Fig.5.7. Thus we have achieved the desired disconnection between displacements and rotations. Originally this element was derived on heuristic principles from a set of stacked serendipity membrane elements but more recent work has shown that it is a hybrid.

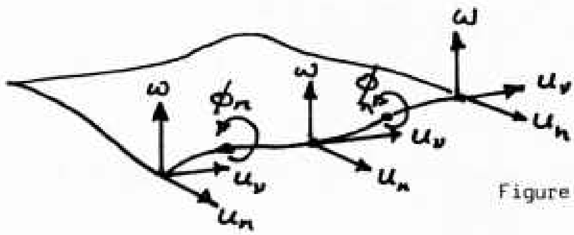


Figure 5.7

An alternative approach similar to that employed with flat plates where a 3-dimensional brick element is reduced down to two dimensional is also used with thin shells. By using reduced integration and other devices an attempt is made to re-create a Kirchoff like shell formulation through heuristic numerical means. This 'hackers' approach to shell elements can sometimes work extremely well but it is fraught with many pit-falls. Once again the importance of validation checks is seen as a central pillar in creating some form of confidence in elements where the internal theory, on which their derivation is based is completely masked by numerical manipulation.

6. CONVERGENCE AND ACCURACY OF RESULTS

We want to be as sure as possible that the procedure we have followed in deriving element and solution methods lead to the correct solution and to this end we want to have some rules to guide us. But, first, we need to assess the kind of solution we are likely to get.

6.1 Equilibrium and Compatibility

For a displacement element we can expect the following:

1) Equilibrium is not usually satisfied inside an element
   - unless we have low grade elements such as constant strain triangles

2) Equilibrium is not satisfied across element boundaries
   - often a good guide to accuracy

3) Equilibrium of nodal forces/moments is satisfied
   - this is an imposed condition

4) Equilibrium is satisfied between applied loads and reactions
   - a useful check as we shall see later

5) Compatibility is satisfied within elements

6) Compatibility between elements across element boundaries should be satisfied
   - although displacement elements have been divided in such a way that inter-element compatibility is violated, this should be avoided if possible.

7) Nodal compatibility is satisfied
- this is an imposed condition

6.2 Convergence

Now that we know what kind of finite element solution we can now expect we can now attempt to list the properties which will ensure that our solution does indeed, converge to the correct answer. That is we wish to be assured that the formulation is such that as we refine the F-E mesh we will monotonically converge to the correct answer at least from an energy view point. In order to achieve this requirement we should satisfy the following conditions:

1) Admissible Shape Functions:

The shape functions which we use on the interior of the finite element must be admissible in the sense defined earlier. Thus they must satisfy the natural boundary conditions of the problem both at structural boundaries and across element boundaries. If the underlying differential equations are of order 2 m in their derivatives the corresponding variation functional (in our case the potential energy) will have derivatives of order m and boundary conditions of order m-1 (i.e. beam $EId^4w/dx^4 = P$, P.E. ⇒ EId^2w/dx^2 boundary conditions = ø (dw/dx) and w).

Thus for an element to handle a structural formulation with derivatives in the differential equation of order 2 m it must have an assumed displacement field with C_m continuous on the interior and be C_{m-1} continuous across element boundaries.

2) Exact Recovery Solutions:

In essence this is a constant strain condition and the element should be able to recover it exactly. For the case of simple elements with simple polynomial displacement fields, such as the constant strain triangle, this fact can be established directly. In the case of more complex elements such as direct evaluation it is not available and we must turn to the 'patch test!'.

3) Satisfaction of Rigid Body Modes

4) Conforming elements - non conforming elements are now being used put they create doubts (possibly conforming in the limit).

5) Geometrically invariant, i.e. has no preferred direction.

6) Polynomials must be complete:

Completeness means that the polynomial contains all the terms up to the specified order.

e.g. a complete linear poly $= a_1 + a_2x + a_3y$

a complete quadratic poly $= a_1 + a_2x + a_3y + a_4xy + a_5x^2 + a_6y^2$

Failure to use complete polynomials often leads to elements which violate conditions (5), i.e. they are biased.

6.3 Accuracy

There are a variety of ways in which a finite element analysis can be persuaded to give inaccurate results, we shall look at two particular aspects.

1) Idealization Error:

In modelling a structure we need to model a real structure which will have joints, bolts, fasteners, local reinforcement etc. When these are modelled the engineer has to make assumptions about the structural behaviour and structural properties. Then he selects the element type, elastic properties, node numbers etc. The resulting F-E analysis will give rise to stresses, displacements etc. which are different from those which would actually occur in the real structure. This sort of general error is known as discretisation error. In certain circumstances we can reduce one of these errors for example in regions of high stress gradients we might want to put in more elements. Curved boundaries are likely to give trouble and require special attention.

Thus certain of these discretisation errors can be removed by careful use of the F-E method and paying attention to detail, but others require testing and correlating with test results.

2) Ill-Conditioning

Ill-conditioning of the stiffness matrix means that changes in the coefficients of the matrix or small changes in the applied loads can cause large changes in the coefficients of the nodal displacement vector obtained from the solution process. This type of ill-conditioning occurs when the terms in the stiffness matrix have large differences in their numerical values - large enough to be influenced by the truncation error inherent in digital computers. The phenomena can also be produced when a region of high stiffness is surrounded by a region of low stiffness. It also occurs in thin shell problems where there is a major numerical difference between bending and membrane strain energies.

In theory, and in practice when serious ill-conditioning problems are anticipated, the condition number of the stiffness matrix can be checked. This is first done by scaling the stiffness matrix with respect to the maximum diagonal coefficient thus

$$\tilde{K}_s = \tilde{S}\tilde{K}\tilde{S} \text{ with } S_{ii} = 1/\sqrt{K_{ii}}$$

where $\tilde{S}$ is a diagonal matrix. The maximum and minimum eigenvalues of $\tilde{K}_s$ are then found (λmax, λmin) and the spectral condition number $C(\tilde{K})$ is then defined as

$$C(\tilde{K}) = \lambda\text{max}/\lambda\text{min}$$

If the computer represents a number with d digits then the results computed are accurate to S digits where,

$$S = d - \log_{10} C(\tilde{K})$$

Thus, for thin shells $C(\tilde{K})$ may equal 10^{12} then

$$S = d - 12$$

and if the word length gives d = 13 then the results are only accurate to one significant figure.

7. NONLINEAR BEHAVIOUR

7.1 Introduction

Our philosophy throughout the course has been to outline the main features of finite elements without giving the details of the theory or methods employed. This is particularly the case with the non-linear analyses of structures which is complex and by no means completely understood. Nevertheless it is an important aspect of the design of structures and emphasis on efficient structures is pushing many engineering disciplines, which have traditionally relies on linear analysis, to consider non-linear behaviour. Non-linearity can occur because the material itself exhibites a non-linear behaviour or because the geometric movement of the structure is large enough to cause cross-coupling between strain fields. Naturally the first of these is termed 'material' non-linearity and the second 'geometric' non-linearity. In the sequal we touch on both aspects.

7.2 Geometric Non-Linearity

In order to keep life simple we illustrate the main theme of geometric non-linearity using the bar element illustrated in Fig.7.1. Here we see a bar originally of length ℓ and resting along the x-axis which is both rotated and distorted to a new length $\ell + d\ell$. The strain in the deformed bar is clearly, $\varepsilon = d\ell/\ell$, the component ε_x is u_x/ℓ but $u_x = u + \delta$.

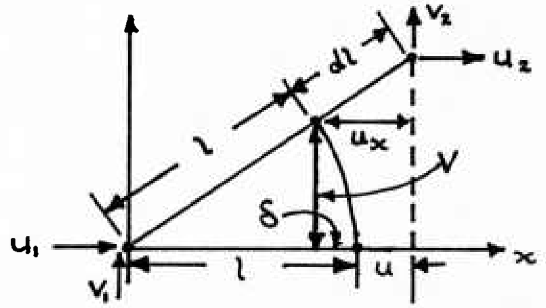


Figure 7.1

If the rotation is not very large then we can compute the value of δ in terms of the displacement V:

$$\ell - \delta = \sqrt{\ell^2 - V^2}$$

$$= \ell\sqrt{1 - \frac{V^2}{\ell^2}}$$

$$= \ell - \frac{1}{2}\frac{V^2}{\ell} + \dots\dots$$

hence $\quad \delta = \frac{1}{2}\frac{V^2}{\ell}$

$$\text{Thus } \varepsilon_x = \frac{u + \frac{1}{2}\left(\frac{V^2}{\ell}\right)}{\ell}$$

$$= \frac{u}{\ell} + \frac{1}{2}\left(\frac{V^2}{\ell^2}\right)$$

and in the limit

$$\varepsilon_x = \frac{du}{dn} + \frac{1}{2}\left(\frac{dV}{dx}\right)^2$$

If we now regard this as one step in an incremental deformation process then the bar will have already been strained (unless this is the first increment). In the general case the strain will, therefore, consist of two parts, the strain accumulated from previous load increments $\varepsilon^o{}_x$ and the new strain increment ε_x thus

$$\varepsilon_x^{\text{total}} = \varepsilon^o{}_x + \varepsilon_x.$$

If we want to move on to computing the stiffness matrix we recall that the material is linear so that we can apply Hookes law and the potential energy term becomes, for a bar element of total length L

$$\pi_p = \int_o^L \left\{ \int_{\varepsilon^o}^{\varepsilon^{\text{final}}} E\varepsilon d\varepsilon \right\} dx - \left[Pu\right]_o^L$$

where $\varepsilon^{\text{final}}$ is the original and incremental strain

P are the loads

u are the incremental displacements

Taking the strain energy term first we have

$$\int_o^L \left\{ \int_{\varepsilon^o}^{\varepsilon^{\text{final}}} E\varepsilon d\varepsilon \right\} dx = E\varepsilon_x{}^2 dx + \int_o^L E\varepsilon_x{}^2 dx + \int_o^L E\varepsilon^o{}_x \varepsilon_x dx$$

and, if we neglect small order terms, this term becomes:

$$= \frac{1}{2}\int_o^L AE\left(\frac{du}{dx}\right)^2 dx + \int_o^L \frac{AE\varepsilon^o{}_x}{2}\left(\frac{dV}{dn}\right)^2 dx + \int_o^L AE\varepsilon^o{}_n \frac{du}{dx} dx$$

where A is the bar cross-sectional area.

Taking the nodal displacements as indicated in Fig.7.1, then our strain energy term becomes:

$$\frac{AE}{2L}\begin{Bmatrix} u_1 & u_2 \end{Bmatrix}\begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} + \frac{AE\varepsilon^o{}_x}{2L}\begin{Bmatrix} V_1 & V_2 \end{Bmatrix}\begin{Bmatrix} 1 & -1 \\ -1 & 1 \end{Bmatrix}\begin{Bmatrix} V_1 \\ V_2 \end{Bmatrix}$$

$$+ \frac{AE\varepsilon^o{}_x}{L}(u_1 - u_2)$$

If we observe that $AE\varepsilon^o{}_n/L = P^o$, the load being applied at the beginning of the increment we note that the last term in the above energy expression is the work done by the initial load moving through the increment of displacement $u_1 - u_2$. We are, therefore, assuming that we start with an applied load P^o with an intial strain $\varepsilon^o x$, finish with an applied load P^{final} with a final strain $\varepsilon^{\text{final}}{}_x$ and u_1, u_2, V_1, V_2 are the <u>increments</u> of displacement.

We now substitute these into the form for πp and the proceed to differentiate this in terms of u_1, u_2, V_1, V_2 we have

$$\underset{\sim}{P} = (\underset{\sim}{K} + \underset{\sim}{K}_1)\, \underset{\sim}{u}$$

$$\text{where } \underset{\sim}{K} = \frac{AE}{L}\begin{Bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{Bmatrix}$$

$$\underset{\sim}{K}_1 = \frac{P^o}{L}\begin{Bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{Bmatrix}$$

$\underset{\sim}{u}$ is the vector of incremental displacements u_1, u_2, V_1, V_2

and $\underset{\sim}{P}$ the vector of incremental loads $(\underset{\sim}{P}^{final} - \underset{\sim}{P}^o)$

The term $\underset{\sim}{K}$ is the normal linear stiffness matrix and $\underset{\sim}{K}_1$ a geometric stiffness matrix which is considered not to vary throughout the load increment. It may be noted that a whole variety of assumptions have been made in obtaining these equations and other interpretations would give rise to alternative geometric stiffness matrices, though the overall principle would remain the same.

The solution procedure requires that the form

$$\underset{\sim}{u} = (\underset{\sim}{K} + \underset{\sim}{K}_1)^{-1}\,\underset{\sim}{P}$$

is solved repeatedly with $\underset{\sim}{K}_1$ reformulated at each step. Thus for a given increment 'i' $\underset{\sim}{u}$ becomes $\underset{\sim}{u}^{(i)}$ the i^{th} increment of the displacement vector $\underset{\sim}{P}$ becomes $\Delta\underset{\sim}{P}^{(i)} = (\underset{\sim}{P}^{(i)} - \underset{\sim}{P}^{i-1})$ the i^{th} increment of the load vector. Hence the full solution to the problem then becomes

$$\underset{\sim}{u}^{total} = \sum_i \underset{\sim}{u}^{(i)}$$

and for the total load

$$\underset{\sim}{P}^{total} = \sum_i \Delta\underset{\sim}{P}^{(i)}$$

This represents the basic approach to the solution of geometrically non-linear structures though the assumptions made in this particular example may be too restrictive for highly non-linear problems. A more general attack using the full Lagrangian strain tensor loads to the form:

$$\underset{\sim}{u}^{(i)} = (\underset{\sim}{K} + \underset{\sim}{K}_1^{(i)} + \underset{\sim}{K}_2^{(i)})^{-1}\,\Delta\underset{\sim}{P}^{(i)}$$

where the stiffness matrices $\underset{\sim}{K}_1$ and $\underset{\sim}{K}_2$ have first and second degree terms in the gradiemts of the incremental displacement vector. Despite this added complexity the philosophy outlined above is clearly preserved. Whichever approach is adopted the numerical solution process is the same and may involve the Newton-Raphson method or some acceptable alternative.

7.3 Material Non-Linearity

The underlying concept behind material non-linearity is that the relationship between stress and strain is non-linear as shown in Fig.7.2. This may be non-linear elastic in which case unloading follows the same curve as that plotted on the loading cycle, i.e. curve A. Or if some 'plastic' deformation takes place then the unloading curve B is different from the loading curve A.

In wishing to follow a curve like A we observe that the stiffness matrix is a function of a parameter which we will take as some form of representative stress σ; though other parameters may be more appropriate to certain classes of problem. Thus we have that the nodal loads on a structure $\underset{\sim}{P}$ are related to the nodal displacements $\underset{\sim}{u}$ by:

$$\underset{\sim}{P} = \underset{\sim}{K}(\sigma)\underset{\sim}{u}$$

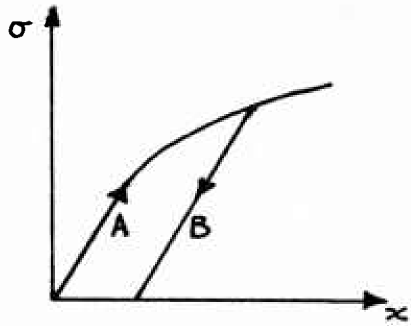


Figure 7.2

For a given final load vector $\underset{\sim}{P}^t$ we shall have need for some form of iterative process to allow us to follow the non-linear behaviour depicted by curve A.

The most straight forward approach is simply to repeatedly solve the equation for given values of stresses thus

$$\underset{\sim}{u}^{(i)} = -\underset{\sim}{k}^{-1}(\sigma^{(i-1)})\ \underset{\sim}{P}$$

Thus we start with the normal linear form for k, solve for $\underset{\sim}{\sigma}$ using $\underset{\sim}{\sigma} = \underset{\sim}{D}\underset{\sim}{B}\underset{\sim}{u}$ then compute $(K(\sigma)$ and re-employ this equation to create a new set of values for σ and so forth. The load applied $\underset{\sim}{P}$ is the final load $\underset{\sim}{P}^t$ which the structure is assumed to carry.

Unfortunately, this simple procedure sometimes fails to converge and a more stable approach is required. A variety of techniques are employed but all work on the basic premise that the stresses used in computing the stiffness should be a more accurate representation of the stress field in the structure for a given step in the iteration. Thus we try to use $k(\sigma^{(i)})$ rather than $k(\sigma^{(i-1)})$ where the unknown $\sigma^{(i)}$ is estimated from the known $\sigma^{(i-1)}$. For example, a first-order Taylor expansion could be used:

$$\underset{\sim}{\sigma}^{(i)} = \underset{\sim}{\sigma}^{(i-1)} + \frac{\partial\underset{\sim}{\sigma}}{\partial\underset{\sim}{P}}\Delta\underset{\sim}{P}$$

This requires that we only increment the load and thus proceed in a step-wise fashion to the final load P^t. In order to fill in the terms of this expression we note that

$$\frac{\partial\underset{\sim}{\sigma}}{\partial\underset{\sim}{P}} = \frac{\partial\underset{\sim}{\sigma}}{\partial\underset{\sim}{u}}\ \frac{\partial\underset{\sim}{u}}{\partial\underset{\sim}{P}} \quad \text{with } \underset{\sim}{\sigma} = \underset{\sim}{D}\underset{\sim}{B}\underset{\sim}{u}$$

now $$\frac{\partial\underset{\sim}{\sigma}}{\partial\underset{\sim}{u}} = \underset{\sim}{D}.\underset{\sim}{B}. \quad \text{and } \frac{\partial\underset{\sim}{u}}{\partial\underset{\sim}{P}} = -k^{-1}$$

so, for $\Delta\underset{\sim}{P} = \underset{\sim}{P}^{(i)} - \underset{\sim}{P}^{(i-1)}$, we have

$$\underset{\sim}{\sigma}^{(i)} = \underset{\sim}{\sigma}^{(i-1)} - \underset{\sim}{D}(\sigma^{(i-1)}\underset{\sim}{B}\underset{\sim}{K}^{-1}(\sigma^{(i-1)})(\underset{\sim}{P}^{(i)} - \underset{\sim}{P}^{(i-1)})$$

and we may now incrementally update the displacements:-

$$(\underset{\sim}{u}^{(i)} - \underset{\sim}{u}^{(i-1)}) = \underset{\sim}{k}^{-1}(\underset{\sim}{\sigma}^{(i)})(\underset{\sim}{P}^{(i)} - \underset{\sim}{P}^{(i-1)})$$

Once again we may start from the linear solution and proceed by employing the above set of equations repeatedly until the final load $\underset{\sim}{P}^t$ is reached.

This broad approach can be used for a range of non-linear applications; plastically, creep etc. In these cases special procedures can be used which take advantage of the specific behaviour being modelled. For example in the case of the elastie-plastic material model the elastic and plastic components can be divided out and treated separately. In this case the computational efficiency and stability of the iterative solution procedures may be improved. However, added complexities occur in realistic cases because certain parts of the structure may be unloading during the incremental loading process. In these situations great care is needed in tracking the loading history if acceptable accuracy is to be achieved.

8. VIBRATION PROBLEMS AND F-E SOLUTIONS

8.1 Introduction

So far we have only considered the solution of statically loaded structures giving rise to responses which do not vary with time. We now want to extend the finite element so that we can be able to deal with structural problems subject to vibrating loads which are very common in aircraft structural design, either as standard vibration problems or as more complex aeroelastic responses.

8.2 Finite Element Formualtion

We begin by defining the Lagrange function L for a dynamical system as

$$L = T - \pi_p$$

where π_p is again the potential energy and T is the kinetic energy. For the moment we omit any dissipative forces. With this function we can now proceed to obtain the appropriate variational principle which allows us to apply the displacement finite element method to dynamical systems. This is called Lagrange's principle and states: Of all possible time histories of displacement states which satisfy the compatibility equations and the constraints or the kinematic boundary conditions and which satisfy the conditions at initial and final times (t_1 and t_2), the history corresponding to the actual solution makes the Lagrangian functional a minimum:

This implies,

$$\delta \int_{t_1}^{t_2} L dt = 0$$

now $T = T(q,\dot{q})$ and, thus, $L = L(q,\dot{q})$ where q is the displacement field associated with the dynamic displacement field within the structure. Thus

$$\delta \int_{t_1}^{t_2} L dt = \int_{t_1}^{t_2} \left\{ \frac{\partial L}{\partial \dot{q}} \delta\dot{q} + \frac{\delta L}{\delta q} \delta q \right\} dt = 0$$

thus

$$\delta \int_{t_1}^{t_2} L dt = \int_{t_1}^{t_2} \left\{ \frac{\partial L}{\partial q} \delta q + \frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}} \delta q\right) - \frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}}\right) \delta q \right\} dt$$

$$= \int_{t_1}^{t_2} \left\{ \frac{\partial L}{\partial q} - \frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}}\right) \right\} dt + \left| \frac{\delta L}{\delta \dot{q}} \delta q \right|_{t_1}^{t_2}$$

$$= 0$$

Thus if we satisfy the initial and final boundary conditions, i.e.

$$\frac{\partial L}{\partial \dot{q}} \partial q = 0 \text{ at } t = t_1 \text{ and when } t = t_2$$

i.e. either

$$\frac{\partial L}{\partial \dot{q}} = \frac{\partial T}{\partial \dot{q}} = m\dot{q} = 0 \text{ or } q = q^* \text{ at } t = t_1 \text{ and } t_2$$

note that $m\dot{q}$ is the momentum.

Assuming that this boundary condition is met then the variational principle demands that q and $\dot{q}$ satisfy the condition

$$\frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}}\right) - \frac{\partial L}{\partial q} = 0$$

If we now introduce a dissipation function R then the principle requires

$$\frac{d}{dt}\left(\frac{\partial L}{\partial \dot{q}}\right) - \frac{\partial L}{\partial q} + \frac{\partial R}{\partial \dot{q}} = 0$$

Taking a specific finite element T and R are given by

$$T = \frac{1}{2}\iiint \rho\dot{\tilde{u}}^T\dot{\tilde{u}}dV$$

$$R = \frac{1}{2}\iiint \mu\dot{\tilde{u}}^T\dot{\tilde{u}}dV$$

and π_p has already been discussed.

Following our usual procedure for a displacement finite element we take a vector of nodal displacement variables $\tilde{d}$ and use the shape function matrix $\tilde{N}$ to give

$$\tilde{u} = \tilde{N}\tilde{d}$$

where $\tilde{N}$ has no components which are functions of time (though 'time' elements are taken in certain fluid formulations). Thus

$$T = \iiint \rho(\tilde{N}\dot{\tilde{d}})^T(\tilde{N}\dot{\tilde{d}})dV$$

$$= \frac{1}{2}\dot{\tilde{d}}^T\tilde{m}\dot{\tilde{d}}$$

$$R = \frac{1}{2}\iiint \mu(\tilde{N}\dot{\tilde{d}})^T(\tilde{N}\dot{\tilde{d}})dV$$

$$= \frac{1}{2}\dot{\tilde{d}}^T\tilde{c}\dot{\tilde{d}}$$

and $\tilde{m}$ and $\tilde{c}$ are called the consistent mass and dissipation matrices, where

$$\tilde{m} = \iiint \rho\tilde{N}^T\tilde{N}dV \text{ and } c = \iiint \mu\tilde{N}^T\tilde{N}dV$$

recalling that at the element level

$$\pi_p = \frac{1}{2}\tilde{d}^T\tilde{k}\tilde{d} - \tilde{d}^T\tilde{p}$$

then the elemental Lagrangian function is given by

$$L_{(e)} = \frac{1}{2}\dot{\tilde{d}}^T\tilde{m}\dot{\tilde{d}} - \frac{1}{2}\tilde{d}^T\tilde{k}\tilde{d} - \tilde{d}^T\tilde{p}$$

Rather than work at the element we assume that we have assembled up the global system so that:

$$L_{(g)} = \frac{1}{2}\dot{\tilde{U}}^T\tilde{M}\dot{\tilde{U}} - \frac{1}{2}\tilde{U}\tilde{K}\tilde{U} - \tilde{U}^T\tilde{P}$$

$$R_{(g)} = \frac{1}{2}\dot{\tilde{U}}\tilde{C}\dot{\tilde{U}}$$

where U are the global nodal displacements, $\tilde{M},\tilde{K},\tilde{C}$, the global mass, stiffness and dissipation matrices with $\tilde{P}$ the global load vector.

Where,

$$\tilde{M} = \sum_{i=1}^{n}\tilde{m}_i, \quad \tilde{K} = \sum_{i=1}^{n}\tilde{k}_i, \quad \tilde{C} = \sum_{i=1}^{n}\tilde{c}_i, \quad \tilde{P} = \sum_{i=1}^{n}\tilde{P}_i$$

for an F-E model having n elements and m nodes

Applying our variational principles at the global level requires

$$\frac{d}{dt}\left(\frac{\partial L_{(g)}}{\partial\dot{\tilde{u}}}\right) - \left(\frac{\partial L_{(g)}}{\partial\tilde{u}}\right) + \left(\frac{\partial R_{(g)}}{\partial\dot{\tilde{u}}}\right) = 0$$

giving $\tilde{M}\ddot{\tilde{U}} + \tilde{K}\tilde{U} + \tilde{C}\dot{\tilde{U}} = \tilde{P}(t)$

If there is no dissipation the matrix C is omitted.

8.3 Mass Matrices

The 'consistent' mass matrix is so called because it is consistent with the stiffness matrix, employing the same shape function. As an example, consider the simpler bar element demonstrated earlier, then the displacement for this element is

$$\tilde{u} = \tilde{N}\tilde{d}$$

where

$$\tilde{N} = \left\{(1 - \tfrac{x}{L})(\tfrac{x}{L})\right\}$$

$$\tilde{d} = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

The consistent mass matrix is then given by the expression

$$m = \iiint \rho\tilde{N}^T\tilde{N}dv$$

$$= \int_0^L \rho A \begin{Bmatrix} (1-{}^x/L) \\ ({}^x/L) \end{Bmatrix} \left\{(1-{}^x/L)({}^x/L)\right\} dl$$

where A is the bar cross-sectional area. Thus

$$\tilde{m} = \frac{\rho AL}{6}\begin{Bmatrix} 2 & 1 \\ 1 & 2 \end{Bmatrix}$$

In the case of the cubic displacement field for the beam bending where

$$N = \left\{\left(1 - \frac{3x^2}{L^2} + \frac{2x^3}{L^3}\right)\left(x - \frac{2x^2}{L} + \frac{x^2}{L^2}\right)\left(\frac{3x^2}{L^2} - \frac{2x^2}{L^2}\right)\left(-\frac{x^2}{L} + \frac{x^3}{L^2}\right)\right\}$$

$$\tilde{m} = \frac{\rho AL}{420}\begin{Bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3l^2 & -22L & 4L^2 \end{Bmatrix}$$

Although the 'consistent' formualtion represents a logical method for generating mass matrices it is not the only approach. An alternative is the so-called lumped mass matrix where a certain amount of structural mass that surrounds a given node is assumed to be concentrated or lumped at that node. In the case where there are both rotational and translational components to the displacement fields the rotational part is sometimes neglected. Whilst the consistent matrix is usually fully populated the lumped matrix is diagonal.

Thus, in the case of the axial bar the total mass is ρAL and the lumped mass matrix is then,

$$m = \frac{AL}{2}\begin{Bmatrix} 1 & 0 \\ 0 & 1 \end{Bmatrix}$$

for the beam the total mass is the same and the lumped matrix

$$\tilde{m} = \frac{\rho AL}{2}\begin{Bmatrix} 1 & & & 0 \\ & {}^L/12 & & \\ & & 1 & \\ 0 & & & {}^L/12 \end{Bmatrix}$$

In this case the rotary inertia about each end of the beam is calculated on the assumption that 1/2 the beam mass is associated with each node. If we ignore rotational terms the mass matrix reduces to

$$\tilde{m} = \frac{\rho AL}{2}\begin{Bmatrix} 1 & & & 0 \\ & 0 & & \\ & & 1 & \\ 0 & & & 0 \end{Bmatrix}$$

which is <u>singular</u>

8.4 <u>Free Vibraitons Analysis</u>

Disturbing an elastic body in the absence of damping and external forces leads to harmonic osciallations. Thus removing these terms from the basic matrix equations give:

$$\tilde{m}\tilde{u} + \tilde{k}\tilde{u} = 0$$

and taking the harmonic solution

$$\tilde{u} = \underline{\tilde{u}}.e^{iwt}$$

leads to the free vibration equation

$$\left\{\tilde{k}-w^2\tilde{M}\right\}\underline{\tilde{u}} = 0$$

where $\underline{\tilde{u}}$ is a vector of the amplitudes of the displacements $\tilde{u}$ and are called the mode shape or eigenvector. The term w represents the natural frequency of vibration and is an eigenvalue for this linear algebraic eigenvalue problem. In this case we do not need to remove the singularity of the stiffness matrix to obtain a solution. Indeed the zero value natural frequencies correspond to rigid body degrees of freedom.

In solving this type of problem we need to set the determinant of the system equal to zero.

$$|\tilde{k}-w^2\tilde{M}| = 0$$

this then gives us the eigenvalues of the system and this is interpreted as the frequency vector w

$$w^T = \left\{w_1, w_2 \ldots\ldots W_N\right\}$$

In order to obtain the mode shapes we observe that the system has n equations to generate the n components of the modal shape vector (or eigenvectors). Thus we select the first element (say) of the displacement vector and set it to unity, thus

$$\begin{Bmatrix} u_1^{(n)} \\ u_2^{(n)} \\ , \\ , \\ , \\ u_N^{(n)} \end{Bmatrix} = \begin{Bmatrix} 1 \\ u_2^{(n)} \\ , \\ , \\ , \\ u_N^{(n)} \end{Bmatrix}$$

Now putting

$$\tilde{E} = \tilde{k}-w^2_n\tilde{M}$$

then $\tilde{E}\tilde{U}^{(n)} = 0$

and partitioning

$$\begin{Bmatrix} E_{11} & E_{10} \\ E_{01} & E_{11} \end{Bmatrix} \begin{Bmatrix} 1 \\ \tilde{u}^{(n)} \end{Bmatrix} = \begin{Bmatrix} 0 \\ \tilde{0} \end{Bmatrix}$$

$$\therefore \quad \tilde{E}_{01}+\tilde{E}_{11}\tilde{u}^{(n)} = 0$$

$$\therefore \quad \tilde{u}^{(n)} = -\tilde{E}_{11}{}^{-1}\tilde{E}_{01}$$

8.5 Eigenvalue Economisers (Guyan Reduction)

Because the problem of solving for a large scale structure is difficult, from a computing viewpoint, it is often found convenient to reduce the size of the eigenvalue problem. The process of reduction involves thinking in terms of 'master' and 'slave' degrees of freedom where the masters are restrained and the slaves removed by condensation.

Although this process does decrease the number of degrees of freedom, and thus, the size of the matrices to be handled it also removes the sparsness of the mass and stiffness matrices. By filling up the associated matrices the process removes the economics of sparsness and this has to be balanced against the advantage of reduced matrix size.

Dividing into 'master' nodal freedoms $\tilde{u}_m$ and 'slave' freedoms $\tilde{u}_s$ gives us a partitioned problem;

$$\left(\begin{Bmatrix} \tilde{K}_{mm} & \tilde{K}_{ms} \\ \tilde{K}_{sm} & \tilde{K}_{ss} \end{Bmatrix} - w^2 \begin{Bmatrix} \tilde{M}_{mm} & M_{ms} \\ M_{sm} & M_{ss} \end{Bmatrix} \right) \begin{Bmatrix} \tilde{u}_m \\ \tilde{u}_s \end{Bmatrix} = \tilde{0}$$

We use the stiffness matrix to define the relationship between the 'master' and 'slave' degrees of freedom which is equivalent to assuming that no loads are applied to 'slave' degrees of freedom in the statics problem, thus,

$$\begin{Bmatrix} \tilde{K}_{mm} & \tilde{K}_{ms} \\ \tilde{K}_{sm} & \tilde{K}_{ss} \end{Bmatrix} \begin{Bmatrix} \tilde{U}_m \\ \tilde{U}_s \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \end{Bmatrix}$$

This gives the required relationship:

$$u_s = -\tilde{K}_{ss}{}^{-1}\tilde{K}_{ms}{}^{T}\tilde{U}_m$$

thus

$$\begin{Bmatrix} \tilde{U}_m \\ \tilde{U}_s \end{Bmatrix} = \tilde{T}\tilde{U}_m = \begin{Bmatrix} \tilde{I} \\ -\tilde{K}_{ss}{}^{-1}\tilde{K}^{T}{}_{ms} \end{Bmatrix} \tilde{U}_m$$

which gives the condensed system

$$\tilde{T}^{T}(\tilde{K}-w^2\tilde{M})\tilde{T}\tilde{U}_m = 0$$

or

$$(\tilde{K}_r-w^2\tilde{M})\tilde{u}_m = 0$$

with

$$\tilde{K}_r = \tilde{T}^{T}\tilde{K}\tilde{T} \; ; \quad \tilde{M}_r = \tilde{T}^{T}\tilde{M}\tilde{T}$$

The transformed matrices K_r and M_r are much denser than original mass and stiffness matrices. The transformed or reduced mass matrix is now a complex combination of stiffness and mass matrices:-

$$\tilde{M}_r = \tilde{M}_{ss} - M_{ms}K_{ss}^{-1}\tilde{K}_{sm} - \tilde{K}_{sm}\tilde{K}_{ss}^{-1}\tilde{M}_{sm}$$
$$+ \tilde{K}_{ss}\tilde{K}_{ss}^{-1}\tilde{M}_{ss}\tilde{K}_{ss}^{-1}\tilde{K}_{sm}$$

The solution to the condensed problem only provides values for $\tilde{u}_m$ and to recover the full vector $\tilde{u}$ we could use the relationship between $\tilde{u}_m$ and $\tilde{u}_s$. However, there will be errors introduced in this process because the slave degrees of freedom $\tilde{u}_s$ are associated with nodes where it is tacitly assumed that these are no loads. Actually, inertia loads should be applied to the $\tilde{u}_s$ nodes while recovering $\tilde{u}$ and to this end we return to the original formulation:

$$\left\{\begin{pmatrix} \tilde{K}_{mm} & \tilde{K}_{ms} \\ \tilde{K}_{sm} & \tilde{K}_{ss} \end{pmatrix} = w^2 \begin{pmatrix} \tilde{M}_{mm} & \tilde{M}_{ms} \\ \tilde{M}_{sm} & \tilde{M}_{ss} \end{pmatrix}\right\} \begin{Bmatrix} \tilde{u}_m \\ \tilde{u}_s \end{Bmatrix} = 0$$

thus

$$\tilde{u}_{sn} = -\left\{\tilde{K}_{ss} - w_n^2\tilde{M}\right\}^{-1}\left\{\tilde{K}^T_{ms} - w_n^2\tilde{M}^T_{ms}\right\}\tilde{u}_{mn}$$

when the eigenvalues w^2_n and eigenvectors u_{mn} for the reduced system are known.

In the major F-E systems, the selection of 'masters' and 'slaves' are automated by scanning the diagonal coefficients of $\tilde{K}$ and $\tilde{M}$ and selecting the first slave for which K_{ii}/M_{ii} is the largest. The matrices are then condensed and the process repeated with the condensed matrices.

8.6 Dynamic Response - Model Analysis (uncoupled equations)

We now turn to the solution of structural problems where the structure is subject to a time dependent applied load. For convenience we shall only deal with undamped structure, though the same arguements apply when certain damping factors are introduced, i.e. Rayleigh damping.

First observe that a general displacement field for a structure can be constructed using the mode shapes for the free-vibration problem. Clearly an approximate solution can be constructed from any suitable function - and the mode shapes have certain properties which we can use with advantage. Thus a general displacement field $\tilde{u}$ is given by

$$\tilde{u} = \sum_{i=1}^{n} \tilde{\varphi}_i z_i = \tilde{\varphi}\tilde{z}$$

where φ_i are the n mode shapes and for a structure with n degrees of freedom and z_i are the modal amplitudes which act as weighting function and are unknown values.

The problem requiring solution is defined by the matrix equation

$$\tilde{M}\ddot{\tilde{u}} + \tilde{K}\tilde{u} = \tilde{P}(t) \quad (\text{with } \tilde{u} = u(t))$$

substituting for $\tilde{u}$ gives,

$$\tilde{M}\tilde{\varphi}\ddot{\tilde{z}} + \tilde{k}\varphi\tilde{z} = \tilde{P}(t)$$

where $\tilde{z} = \tilde{z}(t)$. Pre-multiplying by $\tilde{\varphi}^T$ gives

$$\tilde{\varphi}^T\tilde{M}\tilde{\varphi}\ddot{\tilde{z}} + \tilde{\varphi}^T\tilde{K}\tilde{\varphi}\tilde{z} = \tilde{\varphi}^T\tilde{P}(t)$$

recalling the orthogonality and normality properties of the system we have that

$$\tilde{\varphi}_i^T\tilde{M}\tilde{\varphi}_j = 0 \qquad i=j$$
$$= 1 \qquad i=j$$

$$\tilde{\varphi}_i^T\tilde{K}\tilde{\varphi}_j = 0 \qquad i=j$$
$$w_i^2 \; j=i$$

thus $\tilde{\varphi}^T\tilde{M}\tilde{\varphi} = \tilde{I}$ and $\tilde{\varphi}^T\tilde{K}\tilde{\varphi} = [w^2]$

where $[w^2]$ is a diagonal matrix.

$$\begin{Bmatrix} w_1^2 & & 0 \\ & w_2^2 & \\ 0 & & w_2^2 \end{Bmatrix}$$

Thus we now have

$$\tilde{I}\ddot{\tilde{z}}+[w^2]\tilde{z} = \tilde{\varphi}^T\tilde{P}(t)$$

which may be re-written as

$$\ddot{z}_i+w_i^2z_i = P_i(t)$$

where $P_i(t) = \tilde{\varphi}_i^T\tilde{P}(t)$

The un-coupled equations can then be integrated by one of the direct methods on the computer or solved in some other way.

Because we are only using the modal shapes as approximating functions it may not be necessary to generate many of the actual shapes φ_i - these may also be generated experimentally.

9. FIBRE COMPOSITES

9.1 Introduction

The elements of the theory of finite elements which we have built up in the proceeding sections can be applied to the application of composite fibre structures. In essence there is no problem in modelling this type of material since our theory will admit of anisotropic properties, provided these are accounted for by putting the full anisotropic terms in stress-strain relations. However, in aeronautical applicaitons, it is common to assume that the structure being analysed is adequately represented by a plate modal. In addition, for many applications it is possible to further simplify the model by imposing the Kirchoff hypothesis on the bending terms. This is the procedure which is followed in this short section.

9.2 Plate Model

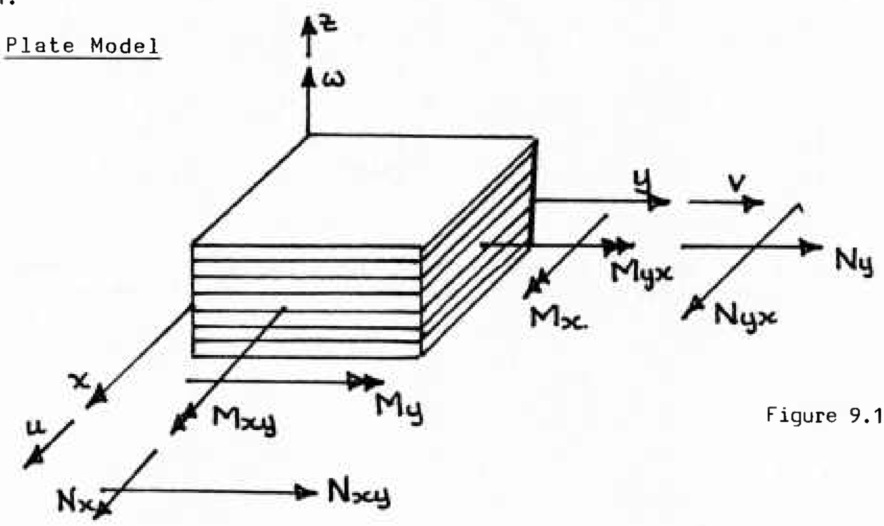


Figure 9.1

The model is assumed to consist of a series of layers of material. Each layer is constructed from a fibre lamina of given orientation (o° to the x-axis, say) held in a matrix of resin. Each lamina is bonded to adjacent laminae by the same resin. This mode of construction means that the whole material is constructed from layers with orthotropic properties. The properties of the layers is computed once the fibre and resin properties are known. The example illustrated in Fig.9.1 has four such orthotropic laminae.

Following our simple palte theory assumption the strain fields given in the laminated structure are:

$$\varepsilon_x = \frac{\partial u}{\partial x} = \frac{\partial u_o}{\partial x} - z\,\frac{\partial^2 w_o}{\partial x^2}$$

$$\varepsilon_y = \frac{\partial v}{\partial y} = \frac{\partial v_o}{\partial y} - z\,\frac{\partial^2 w_o}{\partial y^2}$$

$$\partial_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \frac{\partial u_o}{\partial y} + \frac{\partial v_o}{\partial x} - \partial z\,\frac{\partial^2 w_o}{\partial x \partial y}$$

where u_o, v_o, w_o, represent the two in-plane and the one out-of-plane displacements of the plate mid-plane as illustrated in Fig.9.1. Dividing the strain field into in-plane strains and curvature changes gives:

$$\varepsilon_x = \varepsilon^o{}_x + zK_x$$

$$\varepsilon_y = \varepsilon^o{}_y + zK_y$$

$$\gamma_{xy} = \gamma^o{}_{xy} + zK_{xy}$$

with

$$\varepsilon^o{}_x = \partial u_o/\partial x \quad ; \quad K_x = \partial^2 w_o/\partial x^2$$

$$\varepsilon^o{}_y = \partial v_o/\ y \quad ; \quad K_y = \partial^2 w_o/\partial y^2$$

$$\gamma^o{}_{xy} = \partial u_o/\partial y + \partial V_o/\partial_x \quad ; \quad K_{xy} = -2\partial^2 w_o/\partial x \partial y$$

Recalling that the stress-strain relation is now a more complicated formulation we have

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} Q_{11} & Q_{12} & Q_{13} \\ & Q_{22} & Q_{23} \\ \text{SYM} & & Q_{33} \end{Bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

Thus:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} Q_{11} & Q_{12} & Q_{13} \\ & Q_{22} & Q_{23} \\ \text{SYM} & & Q_{33} \end{Bmatrix} \begin{Bmatrix} \varepsilon^o{}_x \\ \varepsilon^o{}_y \\ \gamma^o{}_{xy} \end{Bmatrix} + z \begin{Bmatrix} Q_{11} & Q_{12} & Q_{13} \\ & Q_{22} & Q_{23} \\ \text{SYM} & & Q_{33} \end{Bmatrix} \begin{Bmatrix} K_x \\ K_y \\ K_{xy} \end{Bmatrix}$$

The stress resultants and moments are defined in the usual way for a standard classical plate theory, but the integration is taken across a series of laminae:-

$$N_x = \int_{-t/2}^{+t/2} \sigma_x \, dz = \sum_{i=1}^{n} \int_{t_{i-1}}^{t_i} \sigma_x d_z$$

$$N_y = \int_{-t/2}^{+t/2} \sigma y dz = \sum_{i=1}^{n} \int_{t_{i-1}}^{t_i} \sigma_y dz$$

$$N_{xy} = \int_{-t/2}^{+t/2} \tau_{xy} dz = \sum_{i=1}^{n} \int_{t_{i-1}}^{t_i} \tau_{xy} dz$$

$$M_x = \int_{-t/2}^{+t/2} \sigma_x zdz = \sum_{i=1}^{n} \int_{t_{i-1}}^{t_i} \sigma_x zdz$$

$$M_y = \int_{-t/2}^{+t/2} \sigma_y zdz = \sum_{i=1}^{n} \int_{t_{i-1}}^{t_i} \sigma_y zdz$$

$$M_{xy} = \int_{-t/2}^{+t/2} \tau_{xy} zdz = \sum_{i=1}^{n} \int_{t_{i=1}}^{t_i} \tau_{xy} zdz.$$

Substituting the definition for σ_x, σ_y, σ_{xy} in these expression and performing the integrations gives:-

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \begin{Bmatrix} A_{11} & A_{12} & A_{13} \\ & A_{22} & A_{23} \\ SYM & & A_{33} \end{Bmatrix} \begin{Bmatrix} \varepsilon^o{}_x \\ \varepsilon^o{}_y \\ \gamma^o{}_{xy} \end{Bmatrix} + \begin{Bmatrix} B_{11} & B_{12} & B_{13} \\ & B_{22} & B_{23} \\ SYM & & B_{33} \end{Bmatrix} \begin{Bmatrix} K_x \\ K_y \\ K_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{Bmatrix} B_{11} & B_{12} & B_{13} \\ & B_{22} & B_{23} \\ SYM & & B_{33} \end{Bmatrix} \begin{Bmatrix} \varepsilon^o{}_x \\ \varepsilon^o{}_y \\ \gamma^o{}_{xy} \end{Bmatrix} + \begin{Bmatrix} D_{11} & D_{12} & D_{13} \\ & D_{22} & D_{23} \\ SYM & & D_{33} \end{Bmatrix} \begin{Bmatrix} K_x \\ K_y \\ K_{xy} \end{Bmatrix}$$

or

$$\begin{Bmatrix} \underset{\sim}{N} \\ M \end{Bmatrix} = \begin{Bmatrix} \underset{\sim}{A} & \underset{\sim}{B} \\ \underset{\sim}{B} & \underset{\sim}{D} \end{Bmatrix} \begin{Bmatrix} \underset{\sim}{\varepsilon} \\ K \end{Bmatrix}$$

Which gives a coupled set of equations relating the stress resultants and moments to the strains and curvature changes.

These may now be used in the potential energy form for a plate containing bending and membrane terms. The appropriate stiffness matrix is then obtained by differentiating with respect to conneciton quantities which will normally ne u_o, v_o, w_o, $\partial w_o/\partial_n$, $\partial w_o/\partial y$ at element nodes. Thus the same problems and potential solutions occur in this type of problem as with the transitional palte discussed earlier.

The added complexity ocver the earlier plate problem arises in constructing the matrices $\underset{\sim}{A}$, $\underset{\sim}{B}$ and $\underset{\sim}{D}$. Many small, micro-based, programs exists to create these terms which may then be fed into the main F-E system. However, some of the main F-E analysis programs do have this capability as part of their pre-processor range.

STATING THE PROBLEM: THE STEP BEFORE F.E. MODELLING

by Ian C. Taig
Chief Engineer, Research
British Aerospace plc, Military Aircraft Division,
Warton, Preston. Lancs. PR4 1AX
England

SUMMARY

Structural analysis is concerned with finding practical solutions of physical problems in the real world; finite element analysis is one powerful tool used as part of this process. Before starting to set up a finite element analysis the task should be planned and the essential features of the real world problem should be identified. In this paper we look at seven steps in this process:- planning the analysis in relation to resources available, definition of the real structure, description of the structural context, statement of the purpose and nature of the analysis, formulation in finite element terms, definition of the facilities and resources available (the solution context) and, finally, prescription of the solution requirements. In all cases, except formulation, these topics are dealt with discursively, without recourse to mathematics. In discussing formulation, a simple "Engineers' Theory" of matrix structural analysis is presented as an everyday medium for defining and understanding the F.E. solution process.

1. Introduction

Finite element analysis of a structure is not an end in itself - but a means to an end. The real objective is always to learn something about the behaviour of a real structure - an assembly, perhaps, of many physical parts made of real, imperfect materials and subjected to real loading conditions which can rarely if ever be precisely known. Boundary and support conditions are those which apply in the real world, whether we are considering a free body in a perturbed airstream, as is typical of a flying vehicle, or foundations in a heterogeneous ground medium for civil engineering structures.

Not only is the primary objective to learn about real structure behaviour, we must also recognise that finite element analysis, whilst undoubtedly the most powerful and universally applicable tool available today, is not the only such tool. It rarely, if ever, addresses all aspects of the real world problem and it is often quite foolish to try to make it do so. For example, it is pointless to use a very fine mesh F.E. analysis to solve the problem of stress concentration around a circular hole in a region of uniform thin plate material under nearly uniform stress - a sound analytical solution exists for that problem which can only be numerically approximated, at considerable expense and effort, by the F.E. method. In almost every such case the better course will be to use F.E. methods to determine characteristic stress levels in the region of the hole and apply analytical or empirical factors to obtain peak stresses.

Again it is pointless to embark on any large or complex analysis without a reasonable appreciation of the size of the task, how long it will take and what it will cost in people and money terms. We would not think of contracting out a job to a bureau without asking for a time and cost estimate, setting deadlines and price limits. Yet how often do we launch an analysis in house without even asking the questions? In my experience, almost every time!

In this lecture, I address the first stages of the analysis process - what might be termed the formal specification of the problem to be solved. This is not only to establish a sound basis for the finite element modelling which is to follow, but also to provide a record so that an independent investigator can follow what it was that the analyst intended. I am talking in non-mathematical terms and addressing the analyst's boss just as much as the analyst.

2. A Methodical Approach to Specification of the Task

The following steps should be followed, either formally or informally, by every structural analyst, before beginning a finite element analysis job. In the U.K., in recent years, the NAFEMS agency has strongly recommended(1) that this process should be formalised and recorded using check lists or proformas. The ideal, in the author's view, (3, 4) is to build in the process as a front end to an F.E. analysis Pre-processor, producing a specification report as output. The suggested steps are:-

- Analysis planning : assuring that right people, data and facilities will be in place at the right time to do the job.

- Definition of the physical structure to be analysed and the sources of authoritative data describing it - especially the physical features of the structure which are considered relevant.

- Definition of the structural context of the analysis: i.e. what other structures or boundary media attach to and interact with the structure, what external and internal loading and temperature conditions apply etc.

- Definition of purpose and nature of the analysis: what kind of results are being sought and for what stage in the design, or approval process are they intended; what kind of solution(s) are required and to what notional accuracy.

- Formulation of the problem in broad finite element terms: how available F.E. solution procedures can be matched to the requirements; how structure loadings, boundary conditions, inertias, constraints can best be represented in broad terms; any special mathematical formulations required for adequate problem representation. One important aspect of problem formulation is treated in some detail later: this is an "Engineers' Theory" of matrix structural analysis.

- Definition of the solution context: i.e. what people and facilities are available, what computing time and/or cost is admissible, above all what is the deadline for delivery of valid answers?

- Definition of solution requirements and presentation i.e. an explicit description of what results should be presented and in what form; in particular, how results should be processed, selected and presented so that users can understand and interpret them.

3. Analysis Planning

Someone, preferably an experienced supervisor, must first decide who is to carry out the analysis and whether that person or team is adequately experienced or qualified to do so. Again, in the U.K., NAFEMS has set out some guidelines(2) to help to establish analyst competence and ways of building up such competence. The analysis team must collectively have adequate understanding both of the structural design or validation problem under study and of the use of finite element methods to solve it. If the available staff do not already meet the requirement, either training or external consultation are necessary, and must be provided.

Incompetent analysis is worse than no analysis!

Data must be available from an authoritative source, at whatever level is appropriate to the status of the project under study. In a large organisation this means having adequate drawings or sketches, loading, temperature and inertia data to a common product standard - all synchronised to fit into the analysis schedule. Other peoples' work must be co-ordinated with that of the stress analyst.

Incompatible data invalidate an analysis before it has begun!

Computing resources must be available on an adequate scale at the time required. This means making an early estimate of the size and scale of the job being undertaken and ensuring that it can, if necessary, be broken down into manageable stages which will fit into computing schedules.

Loss of data or delays from computer overruns are failures of the job planner, not of the computing service.

Special attention may need to be given to the use of automated data preparation and results interpretation facilities. Often there are limitations to their availability: they may use particularly expensive and overloaded equipment or there may only be a few people skilled in their use. Planning must take a realistic view of these issues.

4. Definition of the Structure

In aerospace, it is very unusual for an analysis of a complete vehicle to be performed as one job. In those rare cases, the very complexity of the physical object is such that it is usually necessary to make some restrictive assumptions about the structure to be represented.

In almost all cases, therefore, the analyst must first decide on the physical bounds of the structure to be solved. Usually we break airframes down into major components such as wings, fuselages, empennage structures, etc. or further into structural boxes, fuselage bays, bulkheads, floor structures or such substantial sub-components. Usually these represent physically bounded structural regions - often actually manufactured as individual, self-contained items.

In all but the most local analyses (such as stress distribution in a single detail part or a local region of a component) the real structure will comprise many separate parts, all with finite section dimensions, imperfect intersections, local offsets, gaps, packings and tolerances and the myriad features which characterise real, as opposed to idealised structure.

Drawings and/or computer-based geometry define all the surfaces, datums, detail features and intersections; the real structure definition begins by referencing these sources. Often the structure is being analysed whilst still in its formative stage - the drawings or sketches are undergoing change: it is necessary to identify the standard assumed at the time.

What detail features should be represented will depend primarily upon the analysis purposes: here some judgment or experience may be necessary. Usually there are two questions which may be asked:-

(i) do the analysis purposes require detailed information about stresses or distortions in the immediate vicinity of a physical feature (e.g. stress intensity in the neighbourhood of a particular notch or groove)?

or (ii) is the feature likely to have any noticeable effect on global distribution of stress (e.g. a large lightening hole in an important shear web)?

These questions require an understanding of structural behaviour based on physical rather than on mathematical insight.

At a later stage, a further question may arise:- can I handle all the features I want to represent in a single analysis? This may lead on to a need for substructuring or superelement analysis - a consideration in my next lecture, but it must always be the purpose of the analysis rather than solution expediency which should guide the decision "to represent or not to represent" in the first place. Yet, very often these questions are never even asked - as though their answers were pre-ordained!

5. The Structural Context

Two simple examples will illustrate the importance of the context - i.e. the structural environment in which the component actually performs its function.

5.1 Tension Cleat

One of the simplest, yet most puzzling structural components is the L-shaped cleat or club-foot fitting as shown in fig. 1. Such fittings are widely used, for example, to transmit tension/compression loads from stringers across a transverse diaphragm. To isolate the cleat from its structure and treat it as an independent item under prescribed loadings and boundary conditions as in fig. 1(a) is to ensure disaster. The correct physical boundary conditions can only be determined by taking into account the distortion under load of the members which it connects - both the skin and/or stiffener on the one hand and the bolt, washer and support assembly on the other, as in fig. 1(c). No amount of analytical refinement of the fitting itself as in fig. 1(b), (whether by finite element or any other means) can compensate for failure to place the component IN CONTEXT. Errors of over 100% are commonplace in this type of problem, irrespective of the modelling of the fitting, unless the adjoining structure is adequately represented.

5.2 Multi-hinged Control Surface

A more obvious example of context concerns a multi-hinged structure such as the aileron or flap shown in fig. 2. In its un-deflected position it can often be analysed fairly satisfactorily as though attached to a rigid wing structure, but if the supports are redundant it will always be necessary to take account of hinges and actuating mechanisms, whose flexibilities are comparable with those of the component itself.

When the surface is deflected, as in fig. 2(b), we must either use design devices to alleviate the effects of those components of hinge movement in the plane of the surface (e.g. by use of swinging links) or we must consider the redundant interaction of the deflected surface with the wing as the latter distorts; the aileron may be two orders of magnitude stiffer, when flexing in its plane, as compared with out of plane. A very obvious truth, yet one which most stressmen overlook **once** in their career!

This is an extreme case, but it typifies a situation which characterises most aerospace structures. Whilst we normally analyse components in isolation, all interact with each other at their intersection boundaries and the only correct solutions are those which consider adjoining structures together, subject to loads and distortions which originate on **both** sides of the boundary. Contact problems, where interaction occurs only in compression, with separation in tension , are a special case requiring non-linear treatment or inspired pre-judgment of the solution!

In all cases where components interact at a structurally redundant interface we must perform some kind of substructure interaction, whether we formally set up a full superstructure analysis or use iterative approximations.

6. Purpose and Nature of the Analysis

We have already seen the importance of defining the analysis objective as the essential ingredient in deciding on the representation of structural features. Obviously, the analysis purpose can be significant in many other ways. The whole strategy of modelling may be different if we are considering initial design, detail stress analysis up to ultimate loading or dynamic aeroelastic response within the normal flight envelope. Manageable dynamic or iterative redesign analyses will normally be treated more coarsely than detailed stress analyses. Their loading and inertia representation may need to be the more subtly defined because of this.

In any event a clear statement of purpose is an essential ingredient in a proper definition of the problem, before modelling begins.

7. The Solution Context

Finite element analysis can today be carried out on anything from the humblest desktop micro-computer to a prodigiously powerful supercomputer. The capacity and speed of the machine available and the facilities (such as interactive terminals with pre- and postprocessor software) have an important bearing on how a structural problem should be modelled. Very few problems are intrinsically so complex as to be totally insoluble by even a modest computer - but the amount of ingenuity needed increases as the capacity available decreases. So much so that, for practical purposes, the machine capacity often sets limits on what is normally attempted. In our own case, non-linear analysis is always limited to relatively small and local substructure regions and optimisation is carried out on models with hundreds rather than many thousands of elements - limited by our large mainframe. A supercomputer is needed to extend the practical size limits and might then run into cost problems.

Dynamic analyses, buckling, aeroelastic divergence, flutter, contact problems - all these are examples in which size will be limited to match the facilities available and the time and cost limits.

Sometimes the facilities themselves give a false sense of what is practical. For example, modern mesh-generating programs are so powerful that it is possible to create meshes of vast size and considerable complexity with relatively little effort. Burying a problem in vast numbers of nodes seems easier than thinking out clever modelling devices. It is only when loads, inertias and sometimes even element details have to be defined consistently with the structure that the enormity of the task becomes apparent. We may easily overload any but the largest computing facility and extend the elapsed time for producing evaluated solutions beyond acceptable limits by following the ever-finer-mesh route.

Such judgments can only be made if we are clear from the outset what facilites are available. For example, as already suggested above, loading data preparation is often the biggest single task faced by today's stress analyst. So we need to know what data preparation aids are available for all stages of the analysis and what time and cost limits are acceptable.

In a large company environment, where the mainframe or number-crunching supercomputer is simply there, available as a "free" resource to all who want it, it is easy to slip into careless attitudes where jobs take as long as they take and cost what they cost - as though these were unalterabale facts of life! By any standards this is gross mismanagement both of resources and of timescale.

8. Solution Requirements

To conclude this catalogue of the banal and the obvious, we must get into the habit of stating what we want. Walk around almost any office using number-crunching computing and you see desk tops piled high, cupboards bursting, waste bins overflowing with un-read computer paper. Either that, or rows of zombies in front of screens, searching for the answers they now realise they want and finding they are not available in the form required.

Output requirements should be thought about, written down, and where necessary negotiated, before the job begins, not when it is in its final stages. Selectivity, automated or not, is essential to efficient use of results. The computer can search for worst cases far more efficiently than the user - only when the unexpected happens does the human being really need access to the mass of data.

So <u>think</u> what is needed, <u>state</u> what is needed and provide critera for selection <u>before</u> the event.

9. Formulation of the Problem in Finite Element Terms

When finite element analysis becomes a routine task, formulation of the real problem in F.E. terms often reduces to little more than the nomination of standard solution procedures. If we adopt such an approach, we can set off on a wrong course before the first modelling decision has been made. Very often it is possible to obtain numerical solutions to a problem without ever writing down a single equation - in routine cases this can become the norm. Indeed, the use of matrix algebra to describe the solution of specific problems rarely seems to be taught. Mathematics are used to define the behaviour of elements and to define problem formulation in general terms. But insufficient emphasis is normally given to a number of very simple concepts which place F.E. solution of problems on the same basis as, say, Engineers Bending Theory and the Bredt-Batho thin-walled tube theories.

No conventional stressman would try to understand structural behaviour without recourse to these elementary tools. The same stressman rarely uses the matrix equivalents in thinking about F.E. Analysis and as a result often makes gross errors of judgment in those very areas of structural understanding where his experience is strongest. So before I start addressing any of the details of structural modelling, I would like to go back to first principles and look at some basic structural concepts expressed in the natural mathematics of finite elements, that is **an Engineer's Theory of Matrix Analysis**.

10. Basic Concepts and their Associated Matrix Relationships

There are several basic concepts which underpin the use of the finite element method to solve real physical problems. They translate the classical principles of equilibrium, compatibility, energy minimisation and so on into the language of matrix algebra. We consider:-

- Discrete variables, generalised forces and displacements and correspondence
- Transformation of forces and displacements
- Co-transference, stiffness transformation and kinematic equivalence
- Stiffness matrix reduction and condensation

and conclude with an example illustrating this "Engineers' Theory" in practice.

10.1 Discrete Variables, Generalised Forces and Displacements, Correspondence

The discrete force and displacement variables used in finite element analysis are often regarded as point loads and displacements and in some cases have that significance. More generally they are parameters, associated with physical points, which define force and displacement functions over different parts of the structure. When stiffness matrices are written in terms of these parameters, they are usually expressed in terms of forces and displacements which **correspond** in the normal engineering sense, i.e. the displacements corresponding with a force parameter are those components which do work in association with the force components and vice versa. Some of the common associations of corresponding forces and displacements are shown in fig. 3. In particular, distribution of a force between points corresponds with a weighted average of point displacements, a balancing set of forces corresponds with a relative displacement and interpolated displacements correspond with weighted sums of forces. More subtly, as shown later, the continuous interpolation of displacements corresponds with the integration of weighted body, pressure or line loadings, where the weighting parameters are the same as the interpolation functions. The same line of reasoning carries through to a correspondence between strain - displacement and stress-force relationships.

10.2 Transformation of Forces and Displacements

For the average engineering user, the most important of all concepts to grasp is the simplest - that of transformation. In any problem expressed in terms of discrete variables we may, for convenience, wish to change the variables themselves or their frame of reference. We are all trained to deal with change of axes or change of co-ordinate systems from Cartesian to polar or surface co-ordinates. These involve simple examples of **transformations**: the relationships by which quantities expressed in terms of one set of variables may be transformed to equivalent quantities expressed in terms of a second set. In finite element analysis we are most commonly concerned with linear transformations, i.e. those in which two sets of variables are linearly related to each other. Linear transformations allow us to deal with change of axes, constrained degrees of freedom (simple or complex), symmetry conditions, repeated boundary conditions, rigid body movements, kinematics of mechanisms, reduced basis and modal analyses and many other commonplace analytical situations.

A linear transformation is written, of course, as a simple set of linear equations, expressing the vector of quantities in one frame of reference as the product of a **transformation matrix** times the vector in the other reference frame. Thus a set of displacements U_1 may be related to an initial set U by the equation $U_1 = T_1 U$. Such transformations obey the simple rules of association,
so that if $U_2 = T_2 U_1$ and $U_3 = T_3 U_2$, then
$U_3 = (T_3 T_2 T_1)U = (T_3 T_2)T_1 U = T_3 (T_2 T_1)U$. We cannot, of course, change the order of transformation and only in special cases - square, non-singular or one-for-one transformations - can we invert them. Simple change of axes, using identical numbers of mutually independent variables, is the commonest example of a reversible transformation. If T_1 is such a transformation then $U = T_1^{-1} U_1$.
A rectangular transformation, i.e. one in which the number of variables changes, usually has a great deal of physical significance. Thus if T_2 connects a larger number of variables U_2 to a smaller number U_1 the equation $U_2 = T_2 U_1$ represents (multi-Point) linear constraints because we are saying that all the variables U_2 can be expressed in terms of a smaller number of degrees of freedom U_1. In many cases we can invert such a transformation in terms of a reduced (independent) set of the variables U_2.

Thus if $\begin{bmatrix} U_{2a} \\ U_{2b} \end{bmatrix} = \begin{bmatrix} T_{2a} \\ T_{2b} \end{bmatrix} U_1$ and T_{2a} can be made square and non-singular

Then $U_1 = T_{2a}^{-1} U_{2a}$ and $U_{2b} = T_{2b} T_{2a}^{-1} U_{2a}$

Whence $$\begin{bmatrix} U_{2a} \\ U_{2b} \end{bmatrix} = \begin{bmatrix} I \\ T_{2b} T_{2a}^{-1} \end{bmatrix} U_{2a} \quad \text{or} \quad \left[T_{2b} T_{2a}^{-1} \;\vdots\; -I \right] \begin{bmatrix} U_{2a} \\ U_{2b} \end{bmatrix} = 0 \quad \text{------ (1)}$$

which are constraint equations in standard forms.

On the other hand, if T_3 relates a smaller number of variables U_3 to a larger number U_2, this represents an incomplete transformation. There are now an infinite number of reverse transformations which satisfy the basic equation $U_3 = T_3 U_2$. Particular solutions can be obtained by adding rows to T and dummy variables to U_3 until a square, non-singular matrix $\tilde{T}_3$ is obtained

$$\begin{bmatrix} U_3 \\ U_3' \end{bmatrix} = \begin{bmatrix} T_3 \\ T_3' \end{bmatrix} U_2 \qquad U_2 = \begin{bmatrix} T_3 \\ T_3' \end{bmatrix}^{-1} \begin{bmatrix} U_3 \\ U_3' \end{bmatrix} \quad \text{-------- (2)}$$

10.3 Co-Transference, Stiffness Transformation and Kinematic Equivalence

In conservative systems, the concept of correspondence between forces and displacement can be extended to transformations in what Langefors called "the principle of co-transference." If (r, u) and (R, U) are pairs of corresponding forces and displacements and if u and U are linearly related by the transformation $u = A\,U$, then forces R and r are linearly related by the transpose of the same linear matrix, applied in the reverse direction: i.e. $R = A^T r$. This is a simple consequence of conservation of energy and is not dependent on linear elasticity.

If displacements u and forces r are related by a linear stiffness relation $r = k\,u$ then the principle of co-transference enables us to create the transformed stiffness $\overline{K}$ in terms of R and U.

Thus, $R = A^T\, r = A^T\, K\, U = A^T\, k \quad A\, U$ ---- ③

This familiar result is normally derived directly by minimum strain energy or virtual work arguments, without pausing to state the co-transference principle. In this presentation we suggest that co-transference is the "engineer's equivalent" of a work principle applied to linear algebraic transformations and is valuable for its physical significance. Some simple examples illustrate the principle in action in some common transformation situations.

The rigid link in fig. 4(a) couples points 1 and 2 in the u direction and for small deformations leaves v freedoms uncoupled. We can express the constraint condition by the equation $U_1 = U_2$ and by so doing miss the equivalent force relationship. If we write

$\overline{U} \equiv U_1$ and $U = \begin{bmatrix} U_1 \\ U_2 \end{bmatrix}$, the constraint equation can be written as

$$U = \begin{bmatrix} U_1 \\ U_1 \end{bmatrix} = \begin{bmatrix} 1 \\ 1 \end{bmatrix} \overline{U} \quad \text{or} \quad T\,\overline{U} \text{ where } T = \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

The principle of co-transference tells us that the force $\overline{R}$ corresponding with $\overline{U}$ is given by

$$\overline{R} = T^T X \text{ where } X \equiv \begin{bmatrix} X_1 \\ X_2 \end{bmatrix}$$

The same matrix equations apply if the link is inclined to the (u, v) axes but in this case the vectors U, X, $\overline{U}$, $\overline{X}$ and transformation T take on a modified form.

From fig. 4(b) $U_1 \cos\alpha + V_1 \sin\alpha = U_2 \cos\alpha + V_2 \sin\alpha$

This equation can be used to eliminate, say V_2 in which case

$$V_2 = (U_1 - U_2)\cot\alpha + V_1 = [C \;\; 1 \;\; -C] \begin{bmatrix} U_1 \\ V_1 \\ U_2 \end{bmatrix} \quad \text{-----} ④$$

$$\text{Whence } U \equiv \begin{bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \\ C & 1 & -C \end{bmatrix} \begin{bmatrix} U_1 \\ V_1 \\ U_2 \end{bmatrix}, \text{ where } C \equiv \cot\alpha \quad \text{-----} ⑤$$

This takes the previous form $U = T\,\overline{U}$ if we write

$$T \equiv \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \\ C & 1 & -C \end{bmatrix} \quad \text{and } \overline{U} \equiv \begin{bmatrix} U_1 \\ V_1 \\ U_2 \end{bmatrix}$$

The forces $\overline{R}$ corresponding with the constrained displacements $\overline{U}$ are given by

$$\overline{R} = T^T R \text{ where } R \equiv \begin{bmatrix} X_1 \\ Y_1 \\ X_2 \\ Y_2 \end{bmatrix} \quad \text{-----} ⑥$$

A more graphic example, which also illustrates a second principle, is rotation of axes. From Fig. 4(c) we see that

$$U = \begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix} \begin{bmatrix} u' \\ v' \end{bmatrix} \text{ which is in the same}$$

$$\text{form } U = T\,\overline{U} \text{ if we write } \overline{U} \equiv \begin{bmatrix} u' \\ v' \end{bmatrix} \text{ and } T = \begin{bmatrix} \cos\alpha & -\sin\alpha \\ \sin\alpha & \cos\alpha \end{bmatrix}$$

The transformed forces $\bar{R}$ in inclined axes are given by

$$\bar{R} \equiv \begin{bmatrix} X' \\ Y' \end{bmatrix} = T^T R \equiv \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \begin{bmatrix} X \\ Y \end{bmatrix} \quad \text{-----} \ (7)$$

which is obvious, in any case, by direct resolution. Rotation of axes is a special form of transformation with another important property. In this case we note that, by simple resolution, the reverse force transformation is given by

$$R = T\,\bar{R} \text{ as for displacements}$$

$$\text{Whence } \bar{R} = T^{-1} R \text{ for any vector } R \text{ and hence } T^T \equiv T^{-1},$$

the characteristic property of an orthogonal matrix. Rotational transformations in three dimensions have the same property.

It cannot be sufficiently emphasised that the identity of force and displacement transformations, as seen above, is a property only of orthogonal transformations (particularly axis rotations) whereas co-transference is a universal property of corresponding force and displacement systems.

An important concept, which is easily understood and can be mathematically derived via the principle of co-transference, is that of kinematically equivalent forces. Let us suppose that the displacements v normal to a boundary AB as shown in fig. 5 are defined in terms of a finite number of nodal parameters U. Then $v = A(\xi)\,u$ where $A(\xi)$ is a vector of interpolation functions in terms of the non-dimensional distance ξ along AB. If we divide AB into n equal intervals, the displacements at the centre of each segment are given by:-

$V = A_n\,U$, where A_n is a matrix whose rows are obtained by substituting ξ_n for each segment into vector $A(\xi)$.

Co-transference tells us that the nodal forces R corresponding with U are given by:-

$R = A^T p_n$, where p_n is the n-vector of forces on each segment.

If we write p_i, the force on the i^{th} segment, as $w_i\, l\, \delta\xi_i$ where w_i is the normal load intensity along the edge

$$\text{Then } R = l\,(\textstyle\sum_1^n A_i^T w_i)\,\delta\xi_i \quad \text{-----} \ (8)$$

Which in the limit as $n \to \infty$ becomes

$$R = l\int_0^1 A^T(\xi)\; w(\xi)\; d\xi \quad \text{-----} \ (9)$$

The forces R are described as kinematically equivalent to the continuous line loading, with respect to the displacement functions $A(\xi)$.

For example, if v is a cubic displacement defined in terms of displacements and slopes at A and B we have

$$v = [(2\xi^3 - 3\xi^2 + 1)\quad l(\xi^3 - 2\xi^2 + \xi)\quad (-2\xi^3 + 3\xi^2)\quad l(\xi^3 - \xi^2)] \begin{bmatrix} V_A \\ \theta_A \\ V_B \\ \theta_B \end{bmatrix} \quad \text{-----} \ (10)$$

The nodal forces kinematically equivalent to a uniform line loading w_0 are then, by integration:-

$$R \equiv \begin{bmatrix} Y_A \\ M_A \\ Y_B \\ M_B \end{bmatrix} = \begin{bmatrix} 1/2 \\ 1/12 \\ 1/2 \\ -1/12 \end{bmatrix} w_0\, l \quad \text{-----} \ (11)$$

and those equivalent to a sinusoidal loading $w_0 \sin \Pi\xi$ are

$$R = \begin{bmatrix} 1/\Pi \\ 2l/\Pi^3 \\ 1/\Pi \\ -2l/\Pi^3 \end{bmatrix} w_0\, l \quad \text{----} \ (12)$$

10.4 Stiffness Matrix Reduction and Condensation

In many cases, especially dynamic response analyses, we need to reduce the number of degrees of freedom in a structure for certain purposes whilst retaining the detail for others. We usually use the matrix equivalent of the Rayleigh-Ritz method and express the full set of degrees of freedom as a linear transformation of a reduced set of displacement parameters and derive a reduced stiffness as before. There are many ways of defining the interpolation matrix, relating the detailed degrees of freedom to a coarser set and the selection of the best of these is too complex an issue to be covered here in any detail.

A specialised procedure, often called static condensation, is used in substructure analysis for isolating boundary stiffnesses from those of internal structure. For this purpose we partition a stiffness matrix into regions a remote from and b at the boundary points and write:-

$$K \equiv \begin{bmatrix} K_{aa} & K_{ab} \\ K_{ba} & K_{bb} \end{bmatrix}$$

If a set of forces R_a is applied at the non-boundary stations a we have

$$K_{aa} U_a + K_{ab} U_b = R_a$$

$$U_a = K_{aa}^{-1} (R_a - K_{ab} U_b) \text{ ---- } (13)$$

Whence boundary forces $R_b = K_{ba} U_a + K_{bb} U_b$

$$\text{i.e. } R_b = K_{ba} K_{aa}^{-1} R_a + (K_{bb} - K_{ba} K_{aa}^{-1} K_{ab}) U_b \text{ ---- } (14)$$

The condensed boundary stiffness matrix is given by:

$\bar{K}_{bb} = (K_{bb} - K_{ba} K_{aa}^{-1} K_{ab})$: a familiar formula which is, in fact, expressible as a particular case of matrix reduction as described above. However, this particularly simple reduction is usually quite unsuitable for dynamic analyses because it physically represents the deformation of a structure loaded only at the points in domain b (i.e. under concentrated localised forces) when in dynamic or buckling analyses we need to represent deformations under distributed (e.g. inertia) forces whose resultants are represented by a reduced set of force magnitudes.

Noting the equivalence, in mathematical terms, of corresponding force and displacement transformations, we can choose to define such transformations either by displacement interpolation (deflected shape functions) or by force combinations, whichever seems to be the more appropriate, physically.

Another physically useful concept is to think of static condensation as the inverse of flexibility submatrix extraction. For if the stiffness matrix K is non-singular, its inverse F can be similarly partitioned as

$$F \equiv \begin{bmatrix} F_{aa} & F_{ab} \\ F_{ba} & F_{bb} \end{bmatrix} \quad \text{in which case}$$

$$F_{bb} = (K_{bb} - K_{ba} K_{aa}^{-1} K_{ab})^{-1} = \bar{K}_{bb}^{-1} \qquad \text{or } \bar{K}_{bb} = F_{bb}^{-1} \text{ ----- } (15)$$

We cannot over emphasise the total distinction between static condensation (or its equivalent: flexibility extraction) in which stiffness is expressed in terms of a small number of freedoms with all other freedoms **unconstrained** and stiffness extraction in which all the unselected freedoms are **fully constrained**, usually to zero.

10.5 A Concluding Example of "Engineers Matrix Theory"

When structural assemblies are analysed as separate items, interacting at their boundaries, the analysis is usually treated, in the literature, as a formal substructuring calculation. This assumes that all the substructures are analysed together and solved as one large super-structure problem. More often than not, in practice, the various components will be analysed separately, using preliminary values, sometimes no more than guesses, for the interface loads. Conversion of approximate to accurate solutions and allowance for varying interface geometry (e.g. flap deflection) are straightforwardly handled by our simple theory, whilst often avoided as too complex in standard F.E. texts! Consider, as in fig. 6, a wing initially analysed with prescribed forces R_{oa} at the attachment points (set a) of a flap.

The wing deformations at points a under these prescribed loads (and those over the rest of the wing) are U_{oa}. Define $\bar{K}_{aa}$ as the condensed wing stiffnesses (all other wing nodes unconstrained).

Now suppose we carry out a flap analysis to determine the true set of forces R_a at the boundary.

Additional deformations of the wing ΔU_a are given by

$$\Delta U_a = \bar{K}_{aa} \Delta R_a \equiv \bar{K}_{aa} (R_a - R_{oa}) = U_a - U_{oa}$$

$$\text{Or } U_a = U_{oa} + \bar{K}_{aa} (R_a - R_{oa}) \text{ ----- } (16)$$

Next look at the flap itself, for which we have a stiffness matrix

$$K' \equiv \begin{bmatrix} K'_{aa} & K'_{ab} \\ K'_{ba} & K'_{bb} \end{bmatrix}$$

in local (flap) degrees of freedom - where partition a represents the attachment freedoms to the wing. In any flap configuration the local deflections U_a' are related to U_a by a transformation T which can be written down by inspection of the flap/wing kinematics.

We write $U_a = T\,U_a'$ and correspondingly $(R_a')_w = -T^T R_a$, where the negative sign recognises that the reactions of the wing on the flap $(R_a')_w$ are opposite in sign to the actions of the flap on the wing (R_a). The transformation T is not necessarily orthogonal or even square, non-singular since it may incorporate partial releases such as sliders or swinging links to minimise unwanted interference loadings.

The wing contributes an additional stiffness $K_{waa} = T^T \bar{K}_{aa}\, T$ to the a - partition
If loads $R' = \{R_a'\ R_b'\}$ are applied to the flap we have:-

$$\begin{bmatrix} (K_{waa} + K_{aa}') & K_{ab}' \\ K_{ba}' & K_{bb}' \end{bmatrix} \begin{bmatrix} U_a' \\ U_b' \end{bmatrix} = \begin{bmatrix} R_a' - T^T R_a \\ R_b' \end{bmatrix} \text{-----} (17)$$

From which U_a' may be determined in terms of the, as yet unknown, interaction forces R_a.
Suppose we write the solution in the form $U_a' = U_{oa}' - B\,R_a$
where U_{oa}' contains all the terms involving locally applied loads
Then $U_a = T\,U_a' = T(U_{oa}' - B\,R_a)$
From the wing we have $U_a = U_{oa} + \bar{K}_{aa}\,(R_a - R_{oa})$
So that $(\bar{K}_{aa} + TB)\,R_a = T\,U_{oa}' - U_{oa} + \bar{K}_{aa}\,R_{oa}$ ----- (18)
Whence R_a is determined in terms of the known prior solutions and all the displacements and stresses follow by substitution.

Examples such as this show that there is nothing difficult, either conceptually or mathematically, in formally stating and symbolically solving problems of interacting structures, complicated boundary conditions, internal or external constraints or any of the other practical problems of real structures which often cause difficulties, even for experienced finite element analysts.

We contend that every engineer who is to use finite element methods for problem solving should acquire facility in the use of these simple concepts and techniques so that matrix manipulation of structures should become as familiar as Engineers' Bending, the triangle of forces, Euler buckling and the other standard concepts on which we base our understanding of structural behaviour.

11. References

1. Guidelines to Finite Element Practice. (U.K.) National Agency for Finite Element Methods and Standards, (NAFEMS), Department of Trade and Industry, National Engineering Laboratory, East Kilbride, Scotland August 1984.

2. Guidelines for Assessing Personnel Competence in Organisations Carrying Out Finite Element Analysis. NAFEMS - Presently available to members only - to be published.

3. I.C. Taig, Expert Aids to Finite Element System Applications, in Applications of Artificial Intelligence to Engineering Problems. R. Adey(ed) Springer-Verlag and Computational Mechanics April 1986.

4. I.C. Taig, Expert Aids to Reliable Use of Finite Element Analysis, in Reliability Methods in Engineering Analysis. D.R.J. Owen(ed) University College of Swansea July 1986.

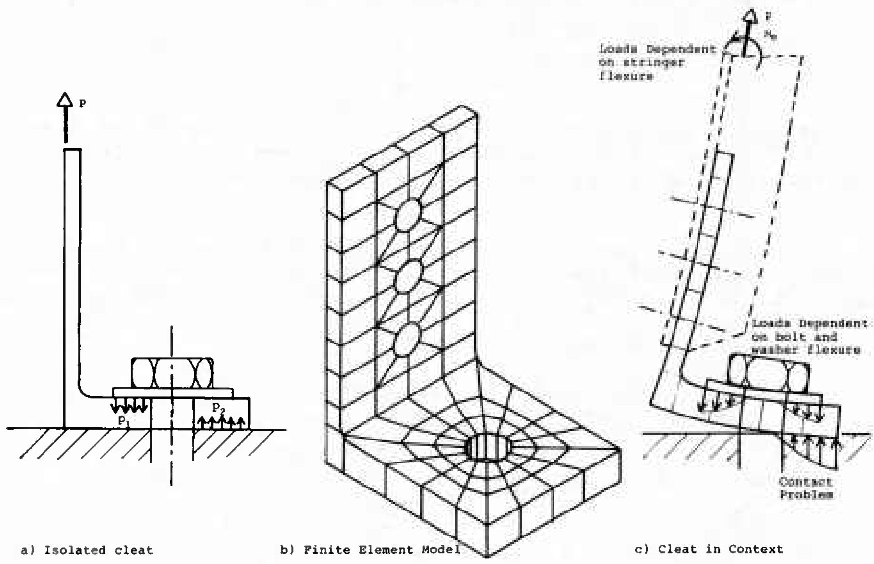


Fig 1 **Analysis of a Tension Cleat**

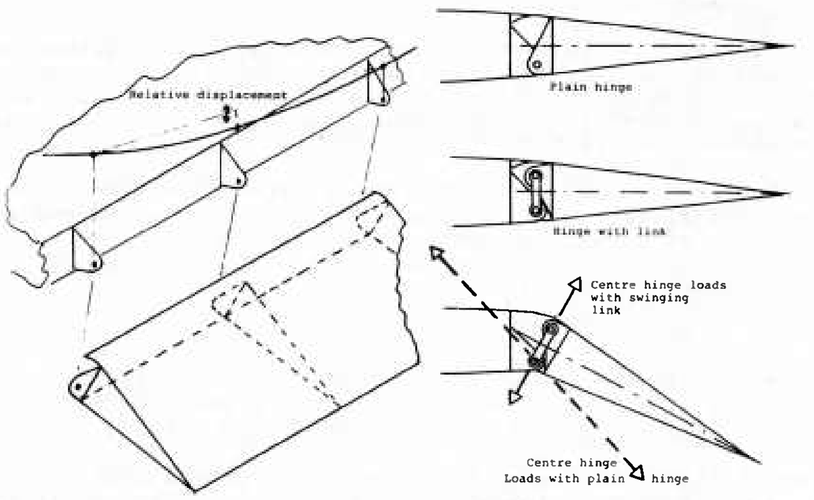


Fig. 2 **Flap-Wing Interaction-Effect of Wing Displacements**

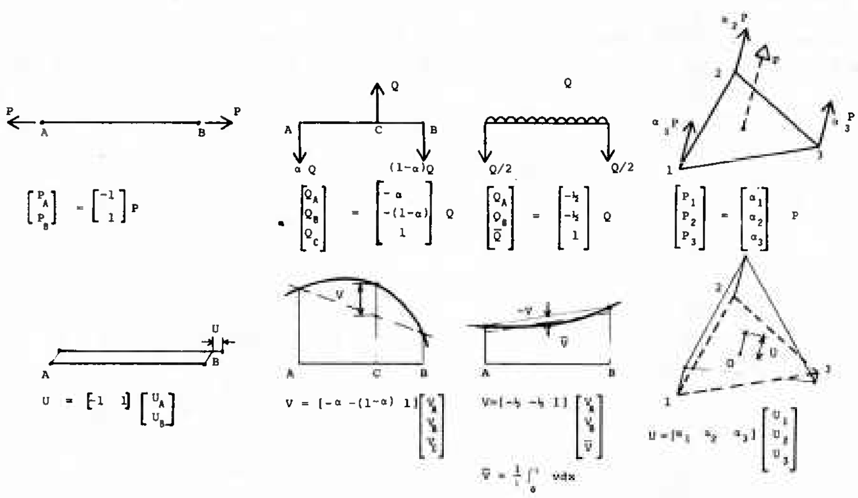
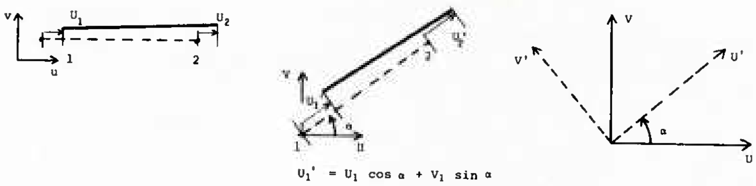


Fig. 3 **Examples of Generalised Forces and Corresponding Displacements**



a) Rigid link (x-wise) b) Rigid link (inclined) c) Rotation of axes

Fig. 4 **Some Simple Displacement Transformations**

Displacement Interpolation for $V_A = 1$

$v = A_A(\xi)\, v_A$ where A is element 1 of vector $A(\xi)$

a) Discrete b) Continuous

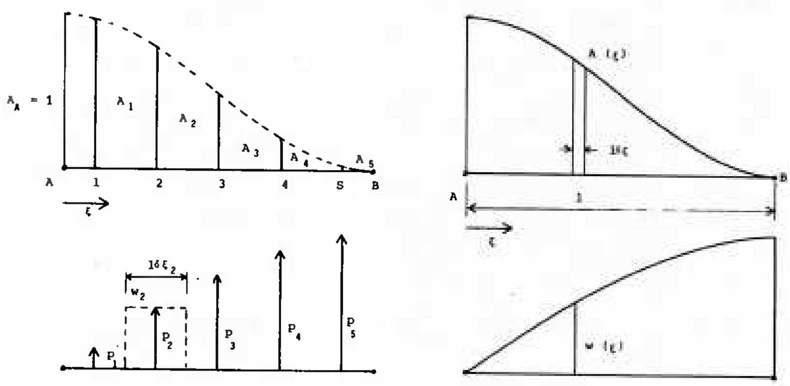
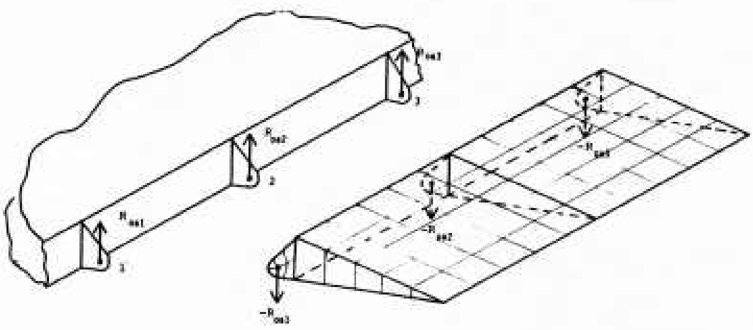
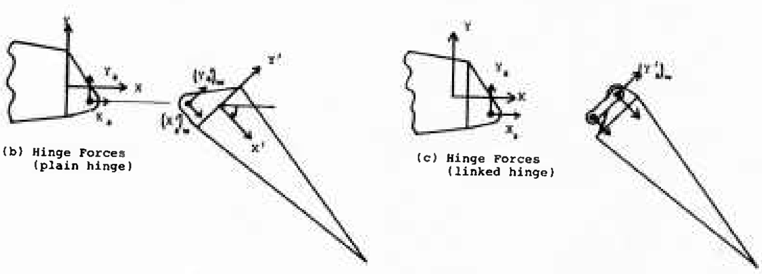


Fig. 5 **Kinematically Equivalent Forces**



(a) Initial (Assumed) Forces at Flap and Wing Hinges



(b) Hinge Forces (plain hinge)

(c) Hinge Forces (linked hinge)

Fig. 6 **Interaction Forces Between Wing and Flap**

MODELLING FOR THE FINITE ELEMENT METHOD

by Ian C. Taig
British Aerospace plc, Military Aircraft Division,
Warton, Preston. Lancs. PR4 1AX
England

SUMMARY

Finite element modelling is not synonymous with mesh generation. The complex structures typical of aerospace, and many other industries, give limited choice for clever mesh definition but contain a host of features which need to be represented, either explicitly or implicitly within the finite element model. Topics covered in this paper include general modelling strategy, definition of a basic mesh, local modelling of structural features inclduing those which are below the basic mesh scale, element selection, load and inertia representation, kinematic constraints and symmetry. All topics are treated in a non-mathematical way, relating decisions which the analyst must make to the known facts about the analysis in the way that experienced people make such judgments. This naturally leads to references to expert systems which are seen as having a major impact in this field.

1. Introduction

Modelling of structures for finite element analysis is the whole activity whereby a real problem in structural analysis is formulated in terms suitable for solution using a finite element computer program. In its Guidelines to Finite Element Practice(1), the U.K. National Agency for Finite Element Methods and Standards makes it clear that modelling is NOT synonymous with mesh generation, as much of the literature on the subject would suggest. Determination of the appropriate mesh is but one aspect of the problem: many others are equally important and often give the analyst more difficulty. Likewise, accuracy of the finite element solution, per se, is only one of several criteria which influence modelling decisions.

The growing body of literature on mesh generation, optimum gradation and adaptive refinement addresses an important issue in relation to F.E. analysis of continuum structure, in which the complex boundary and the (homogeneous) material properties characterise the structure. In aerospace structures this is rarely more than a small aspect of the problem facing the designer and stress Engineer(2, 3); sometimes it is completely irrelevant. In addressing the aerospace engineer, we start from the fundamental position that our structures are so complex that the lowest level of significant structural detail (e.g. small holes, cracks, grooves, fillets etc.) is well below the scale at which a single comprehensive finite element model can explicitly represent it. We think from the outset in terms of multi-level analysis (ranging from global to local) whether formally interrelated by sub-structuring or not. The predominant issue at the global representation level is the extent to which local features should be represented explicitly at all - not how the mesh should be best refined to provide accuracy in their vicinity. At the local level, mesh refinement is a major issue, but so too is structural context (i.e. all aspects of the interfaces with adjoining structure). Even at this scale, actual features such as discrete fasteners, composite lamination boundaries, fillets and lands can dictate practical, as opposed to optimal, mesh definiton.

At all levels we are concerned about boundary conditions, load and inertia definition, realistic material behaviour etc., all to be represented at a cost and within a timescale we can afford as well as to give the accuracy we desire. Many of the decisions to be made are heuristic, i.e. based on judgment and experience, rather than mathematical or algorithmic. This is why they have received such scant attention in the scientific literature: a situation which is starting to change now that expert systems are becoming practically feasible and scientifically respectable(5).

We attempt here to give a broad coverage of the subject by looking briefly at the following stages in modelling:-

- High level structural representation (the generic types of elements to be used) and modelling strategy
- Basic mesh definition to represent the primary characterisitcs of the structure at an appropriate level of definition
- Local modelling of structural features, via explicit fine meshing, special elements or implicit representation in modified element behaviour
- Specific element selection; shapes, formulations and material properties
- Representation of local and distributed loads and inertias
- Kinematic boundary conditions: symmetry and constraints.

2. Structural Representation and Modelling Strategy

We begin by restricting our discussion to the structures commonly associated with aerospace: reinforced shells of great complexity, beam and framed structures and solid fittings of complex geometry. We assume that the "real world" problem is adequately defined as recommended in the previous paper(4). The first question to be addressed is what generic type of structural representation is appropriate: often the answers seem so obvious that the question is not asked, but we should take the time, occasionally, to to review our standard practices. Some high level issues which may arise are outlined below.

2.1 Shells, Plates or Solids?

For an aircraft fuselage or an undercarriage mounting bracket the answer to this question is obvious. But what of the solid missile wing, the undercarriage oleo leg, the thick skin at the root of a composite wing, the one-piece forged or machined airbrake? For such structures a balanced judgment is needed, weighing the analysis purposes and accuracy on the one hand against cost and complexity on the other. Some of the considerations affecting the decision are set out in Table 1 below.

TABLE 1

Choice of Shell or Solid Representation

For Shell	For Solid
° Structure is of characteristic shell or plate form or built up from such members ° Thicknesses small compared with other significant dimensions ° Mainly interested in stresses at extreme fibres or in displacements ° Homogeneous, isotropic material ° Through-thickness or out-of plane shear stresses unlikely to be significant ° Cost or job size are serious limiting factors ° Adequately proven solid elements are not available ° No efficient shell-solid transition elements are available	° Structure is of general 3-dimensional form ° Thicknesses are significant compared with other relevant dimensions ° Stress distribution through the thickness is likely to be important, especially when there are stress raisers in the depthwise direction ° Strength through the thickness or in out-of-plane shear is lower than in-plane ° Progressive non-linearity may develop, moving in from the surface ° Specialised solid elements (e.g. orthotropic sandwich core) provide efficient representation of densely - packed structure or quasi-solid material

Similar questions arise in frame and beam-type structures where we must decide on line elements versus representatively modelled sections. In all cases, we recommend that, other things being equal, the simplest representation is the best!

2.2 Membranes or Plate/Shells; Facets or Curvature

At first sight it may seem pointless to use facet membrane representation of any curved shell when there appear to be plenty of shell elements available. But not only are there order-of-magnitude size and cost differences, there is also no such thing as an impeccable curved shell element, which can be reliably used (and interpreted by ordinary mortals) without significant added complexity. One need only cite the old problem of the rotation component normal to the shell surface (and the many dubious practices used to suppress or circumvent it) to realise the practical difficulties.

In academic circles, the use of lower order elements (e.g. linear isoparametrics) is almost unknown, yet in industry, where complex built-up structures are being analysed, these are still the norm. There are good reasons for this, despite the readily available evidence that "higher-order is better" from a cost: accuracy viewpoint. Linear elements are both easy to use and understand and are reliable (if not precise) in performance. They can be safely used in conjunction with statically equivalent loads without needing to bring in the added complexities of kinematic equivalence. Furthermore, if the structure (at the scale of the F.E. analysis mesh) is not a continuum, but an assembly (or intersecting network) of discrete members, then all theoretical advantages of higher order elements can be lost.

The issues which affect these high level representation arguments were explored in some detail in a recent expert systems project(5). For example, Fig. 1 shows a so-called called inference net, taken from ref. 5 which indicates how a recommendation to use membrane (as opposed to flexural) representation was derived.

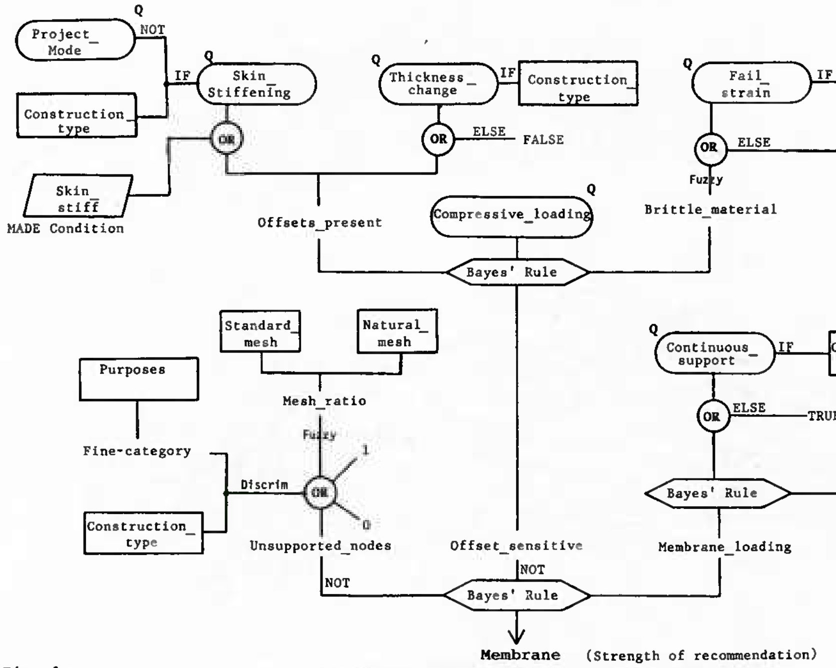


Fig. 1 Derivation of the Membrane Recommendation

Without going into detail, the diagram shows that three factors influence the strength of recommendation in favour of membrane representation:- (the predominance of) membrane loading supports the recommendation whilst (the presence of many) unsupported nodes and (the probability that the skin is) offset sensitive oppose it. Bayes' rule provides a means of combining and weighting these factors, to emulate balanced human judgment. The three influencing factors are themselves derived from other considerations such as analysis purposes or (the use of) brittle material, ultimately leading back to direct questions to be asked of the analyst, such as (is there significant) normal pressure (not reacted by circular curvature or closely spaced supports)? The judgments are seen to be complex but they lead back to questions which are individually quite simple. The expert systems methodology provides the first practical way of making such judgments, and their associated questions, readily accessible to people without wading through impossibly - complex manuals.

2.3 Symmetry or Repeated Structures

As a general rule, if symmetry is present in a structure, take advantage of it to reduce the problem size; likewise if structural patterns (and their boundary interactions) are repeated. In some analysis systems, this poses no practical problems but in others it can give rise to a great deal of complexity. For example, a fuselage structure may be structurally symmetric over part of its length and asymmetric over another part. Whether or not we take advantage of symmetry, where it exists, depends upon how readily we can combine symmetric halves (and their, usually, asymmetric loadings) with the asymmetric portion of the shell to obtain the overall solution. Often the complexity of this task, to say nothing of the false starts and re-runs, makes the alternative - a doubled size for part of the analysis - look very appealing!

If fully automated and clearly explained facilities are not available in the finite element system being used, it is foolish to enter this minefield without first acquiring some manipulative skill in symbolic problem-solving using the "engineers' theory" of matrix analysis of the previous paper(4) or its equivalent. It requires some skill and experience just to get signs right and to avoid errors by factors of 2!

We always recommend that trial runs on small problems be carried out and validated before attempting large analyses. Time must be allocated for building up this confidence - it will usually be repaid in fewer abortive runs.

2.4 Modelling Strategy

We have already touched upon some of the stratigic issues which must be settled at the very start of modelling. We now look at some of these more systematically.

° Mesh scale in relation to features

Many aerospace structures are designed and built as roughly rectangular assemblies of skins, supporting members, stiffeners and so on. The pattern of actual members dictates a (possibly more than one) **natural mesh** scale, simply by following structure intersections and improving proportions by regular sub division of slender panels.

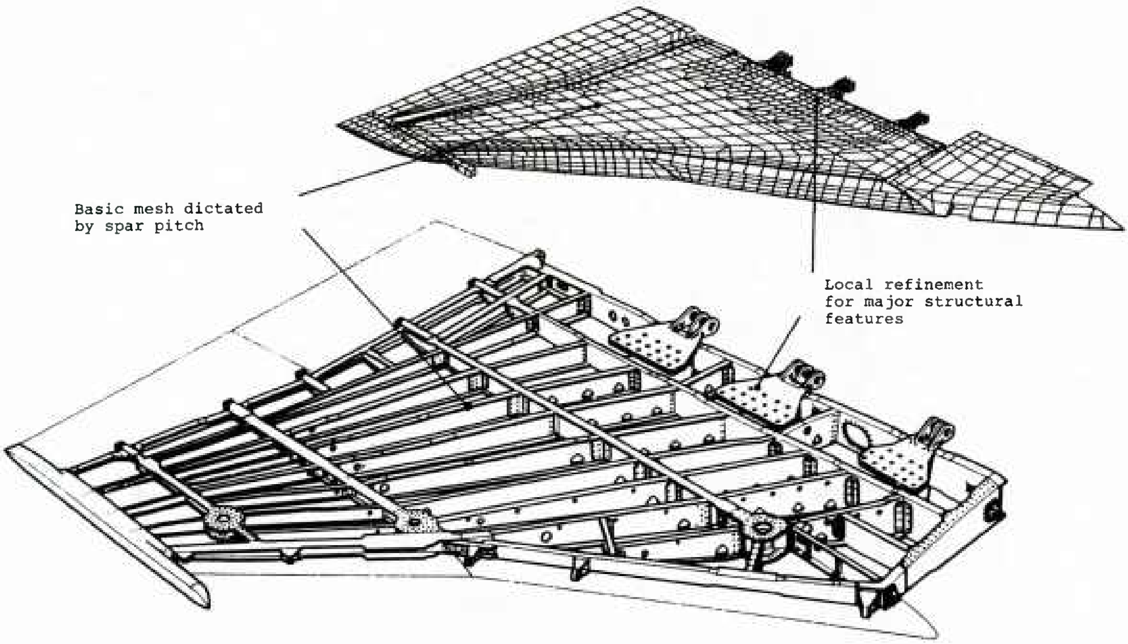


Fig. 2 **Wing Internal Structure and Design Analysis Mesh**

The wing shown in fig. 2 is an example of a structure whose natural mesh is clearly dictated by the closely spaced spars. Whether this natural mesh is not fine enough, just right or too fine for general analysis is a question which must first be related to the analysis purposes and perhaps modified by size and cost considerations. With today's company computers it is rarely necessary to go coarser than the natural grid defined by the major supporting members (spars, ribs, frames, bulkhead etc.) but equally rarely feasible to get down to individual skin stiffeners for closely-pitched stiffened skins. It is almost never feasible to represent skin stiffeners in full section detail in a single pass analysis. We showed in a recent AGARD discussion paper that such an analysis might require 10^5 nodes x 6 dof. per node for a wing, more for a fuselage - a huge task for multi-case, multi-purpose analysis, for pre- and post-processing and for information handling.

° Super-modelling and Sub-modelling

A totally different approach to modelling is appropriate if we choose a basic mesh scale much finer than the natural mesh as opposed to a similar or coarser scale. In the former case, which I shall subsequently term **super-modelling** we can base decisions on the continuum analysis reasoning favoured by the academic community. In the opposite case, hereafter called **sub-modelling**, continuum strategies are largely irrelevant. As suggested above, most major airframe components will fall into the sub-modelling category. Super-modelling will be confined to local regions and isolated structural members. The divergence in strategy is indicated in Table 2 below.

TABLE 2
Indications for modelling strategy related to mesh scale

Super-modelling (more mesh than features)	Sub-modelling (more features than natural mesh)
° Graded meshes refining in regions of geometric or load-induced stress concentraion ° High order elements indicated ° **Explicit** representation of the relatively few major structural members with mesh density appropriate to significance ° Stress gradients, principal stress contours, energy densities guide mesh definition	° Meshes in multiples or sub-multiples of basic mesh, determined by structure ° Simple, low order elements indicated ° Minimal (single line or panel) representation of major skin-support members ° Mesh refinement, in local sub-multiples of basic mesh, only at most significant stress raisers (continued over page)

TABLE 2 (continued)

Super-modelling (more mesh than features)	Sub-modelling (more features than natural mesh)
° Accuracy principal criterion ° Adaptive refinement and post-analysis correction are appropriate	° Sub-scale features normally **implicit** in standard mesh elements; representation by subsidiary analysis, special elements and sub-structuring ° Convenience, size, cost the principal criteria

Substructuring and Superelements

Substructuring is a device for breaking down an analysis into manageable parts. Only if there are several identical and repeated substructures does it offer any savings in either elapsed time or solution cost. Usually it adds a great deal of complexity to the analysis specification and actually increases the overall job size. We must distinguish between substructuring which is only recommended when there are sufficient supporting reasons, from partitioned data preparation, which breaks down the modelling process but not the solution into manageable and natural parts; this is recommended for **all** large and fairly complex structures.

Reasons which support substructuring, in roughly descending order of importance are:-

° Relative geometry of adjoining structures changes (swing wings, deflecting control surfaces, etc. etc.)

° Different analysis teams assigned to tasks.

° Single-pass analysis exceeds available computing time slot (may be better to try a bigger computer!)

° Localised iterations required, e.g. non-linear analysis, contact problems etc.

° Several components very sparsely connected (higher priority if nearly statically determinate)

° Multiple, repeating substructures (but probably better to use modified stiffness matrix assembly)

Remember that substructuring, taken through to rigorous solutions, complicates structure and load definition, solution (especially for dynamic and iterative solutions) and verification. More errors are likely and there are no pre- and post-processing advantages compared with simple data partitioning.

When substructuring is used it is often necessary to analyse substructures independently with approximate (e.g. coarse mesh) interface conditions and only to carry out full interaction if there is reason to suspect that the interface conditions are causing local disturbances in stress patterns.

There is a special form of substructure analysis, involving generalised prior solutions of standard components, considered later under the heading of implicit modelling.

Another simplifying device is to analyse smaller structures whose boundaries overlap so that St. Venant's principle can be allowed to do its work and we make two estimates of behaviour in the overlap region.

3. Basic Mesh Definition

Having discussed some of the strategic issues which depend upon the mesh scale in relation to real structure, let us now return to the mesh definition in a little more detail.

It is implied in the nature of most geometric pre-processor systems that structures to be analysed comprise components bounded by a relatively small number of continuous surfaces. It is predicated that the analyst's problem is to create a mesh by interpolation between the relatively coarse defining boundaries and that there is freedom to select intermediate mesh lines for convenience and accuracy. As we have seen already, this situation can apply when we are able to adopt super-modelling strategy, i.e. when dealing with relatively, small, self contained components. For the large majority of aerospace structures we are in a sub-modelling situation, where the defining boundaries of the structure are already very complex and, whatever scale we choose for analysis, there will usually be significant structure features at a smaller scale. Only in very local regions, where significant features are sparse, do we have the luxury of free choice of intermediate mesh lines. At the most we deal with standardised ways of blending a local mesh (such as a ring of nodes around a large hole) into a more-or-less rectangular pattern.

To formalise the discussion, we consider mesh definition in two parts:- i) creation of a **basic mesh** pattern which is directly related to the pattern of structural members and where the principal considerations are scale of the mesh and proportions of the elements

ii) modification of the basic mesh to account for special structural features - we term this the **local mesh** and will consider it further in the next section.

3.1 How Fine is Fine?

We normally describe a mesh as **coarse** or **fine** and naturally associate these qualitative descriptions with the purposes of the analysis. Whilst these terms are self explanatory to the experienced analyst, they need some numerical interpretation for guidance of the relative novice; this depends on the size and complexity of the structure being analysed. For structures like aircraft wings and fuselages, coarse usually means representing all major intersecting members and doing no more than making a few integral sub divisions of natural bays of high aspect ratio. The **natural mesh** thus defined in fig. 2 would therefore be described as coarse. Where there are few intersecting members, for example in a sandwich core-filled flap as shown in fig. 3, a coarse mesh is defined by the minimum number of stations needed to give reasonable stress and displacement patterns across the lesser plan form dimension. Five or six stations across the width, corresponding pitch along the length and no additional stations through the thickness will be most peoples' idea of a coarse mesh.

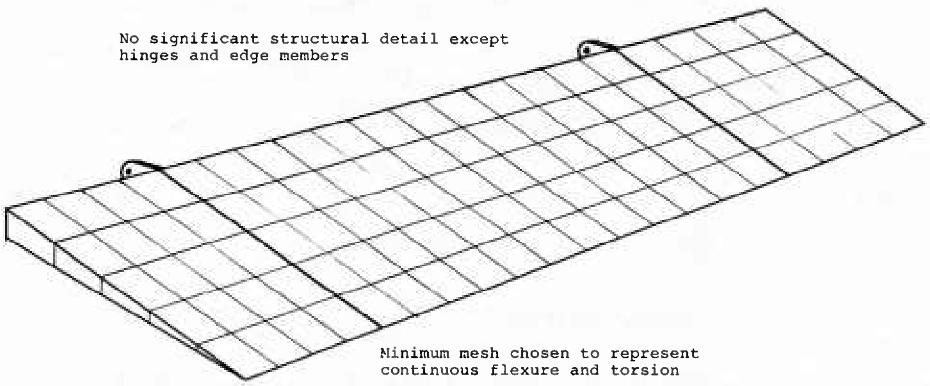


Fig. 3 Minimum Mesh for a Simple Flap

In the expert system described in, ref. 5, the following formula is used (in a "fuzzy" sense), for topologically cylindrical shells, to define the coarsest or **minimum mesh** size for a shell with few intersecting members.

$$\text{Minimum mesh size} \simeq 50 \times \frac{(1 + 0.6 \times \text{Depth-ratio}) \times \text{Aspect-ratio}}{\text{Symmetry-factor}}$$

where the symmetry factor is 1.8 for single symmetry, 3.5 for double symmetry and 1 for no symmetry.

The flap of fig. 3 with no symmetry, aspect ratio 4 and depth ratio 0.1 evaluates to 212 nodes which compares with the 231 nodes shown for 6 chordwise stations. The formula applies to within about 20% for a wide variety of shell cross sections, but needs modification for multi-surface shells (e.g. fuselage with internal floors, walls or engine tunnels and ducts).

Returning to the more complex structures such as fig. 2, the decision whether or not we need a basic mesh finer than the natural one depends on our assessment of "how fine is the natural mesh already?" A measure of this is the **natural mesh ratio** defined as $\frac{\text{Natural mesh}}{\text{Minimum mesh}}$. To complete the formal description of the decision process we have categorised analysis purposes into three broad groups:-

- **Coarse purposes** are those which create no special demands for refinement beyond the natural mesh

- **Fine purposes** are those which may require refinement dependent upon the magnitude of the natural mesh ratio

° **Very fine purposes** are those which certainly require a finer level of detail than that provided by the natural mesh.

Refinement in this context means integral sub division of natural mesh pitches and maintenance of element aspect ratios in an acceptable range (we normally use ½ to 2 for skins, up to 5 for internal webs, with "near square" the preferred shape). Examples of categorisations which we have adopted (as general guidelines subject to override by the analyst) are tabulated below.

TABLE 3
Categorisation of Analysis Purposes

Coarse Purposes	Fine Purposes	Very Fine Purposes
Initial sizing of structure Global optimisation Global flexibility determination Static aeroelastics Global dynamic response Dynamic stability (flutter)	Design stress analysis Global static stability Post-buckled, quasi-static response Damage tolerance analysis Thermal response Global + local dynamic response Panel flutter	Check stress analysis Fracture mechanics Laminated structure analysis Global + local stability Acoustic response

Combining all these factors with others pertinent to particular tasks makes the ultimate decision complex and subjective. But in the end we came down to a relatively simple choice: "one, two, three or more sub divisions of the natural mesh, or none at all?" Numerical assessments are possible but they can only be approximate, within rather large tolerances. One way of dealing with this, in expert systems design, is to give weighted recommendations based on formulae and fuzzy set theory so that we might advise the user (based on information on purposes and knowledge of the natural mesh proportions):-

No sub-division	:-	0.07	Strengths of recommendation in range 0-1
1 sub-division	:-	0.56	
2 sub-divisions	:-	0.32	
3 sub-divisions	:-	0.05	

making it a matter of personal preference or convenience which option to adopt. We use this approach frequently in our FEASA system(5).

3.2 Graded Meshes

The previous discussion relates to a "typical" mesh which is implied to apply fairly uniformly over a structure. If we are in the sub-modelling situation (with more features than mesh points) this fairly uniform coverage is usually appropriate. But in situations like the flap in fig. 3 it is obvious that the major load concentrations and localised deformations will occur in the immediate vicinity of the hinges. The uniform mesh of fig. 3 is clearly not ideal and we are faced with two basic options: a) treat the hinge zones as **local features** and use local sub division and blending techniques as discussed later in section 4.

b) use a gradually **graded mesh** in which a smooth transition is made between the uniform mesh remote from the hinges and the local mesh defined by the hinge geometry and attachment features.

Mesh gradation is a topic in its own right with an extensive and often esoteric literature. Basically there are two distinct approaches, namely **a priori** methods, in which we base decisions on knowledge available before analysis, and a **a posteriori** methods in which we use the results of one or more analyses to improve a trial mesh. The latter topic leads on to self-adaptive mesh refinement, hierarchical elements and many other topics outside the scope of this lecture (see, for example, Babuška(6), Zienkiewicz(7) and Brandt(8)). A priori methods rely heavily on experience to know how fine to make the local node spacing and how to blend into the remote mesh. Often, if automatic mesh generators are used, the options for blending are very limtied. Regions of structure must be sub divided into combinations of topological rectangles and/or triangles with a limited range of possibilities for specifying interpolation parameters. In arriving at a judgment, the practical engineer will take into account many factors in addition to accuracy, for example:-

- Profiles of thickness changes or laminations
- expected principal stress trajectories
- mesh continuity for ease of presentation and interpolation
- maintaining sound element proportions
- picking up sub-scale features such as bolt holes

My own preference is to keep to near-square quadrilaterals and if necessary to sub-divide without blending, by using constraints on boundaries where nodes are incompatible. Standard blending patterns, such as from squares to circles with approximately geometric progression in mesh spacing, fit well with use of automation.

4. Local Modelling of Structural Features

This is perhaps the most important aspect of the whole modelling topic for the aerospace engineer. Knowing what to represent, how to represent it and in how much detail is the art which separates the expert from the novice, the good analyst from the bad. In the previous paper we emphasised that it is pointless to try to achieve more than finite element analysis can deliver. Stress concentrations at tiny holes, crack tips and so on are best dealt with by appropriate local analysis, which is not to say that the products of such analysis should not be incorporated implicitly in the F.E. model. What we want to avoid is the pursuit of indefinitely fine meshing in regions of near singularity.

In a recent paper(9) we introduced (or more accurately gave a respectable name to) implicit modelling and implicit mesh refinement as the pragmatic alternatives to explicit fine meshing and comprehensive substructure analysis. They are, in fact,concepts which have been embodied from the very first day that finite element analysis was used in anger, yet this may be the first attempt to give them systematic consideration. The first issue in local modelling is thus to answer the following question.

4.1 Explicit or Implicit Modelling?

We are now looking at **sub-scale features** of the structure, i.e. structurally significant features whose defining geometry is at a smaller scale than the basic grid, previously discussed, and at **major features** which perturb the basic grid, such as cut-outs and doors in otherwise continuous skins. The alternative approaches are:-

° **Explicit modelling** in which we represent the important geometry of the features by appropriate local mesh and element selection and blend in to the basic mesh

° **Implicit modelling,** in which we continue the basic mesh with no more than minor changes (duplication of occasional nodes to allow flexible coupling is as far as we go) but modify the section properties of elements and/or their materials so as to simulate the effect of the features as seen by adjoining structure. When using implicit modelling it is assumed that the analysis will yield good boundary conditions to enable a subsequent local analysis of the features to be performed. Implicit modelling is thus a hybrid form of substructure analysis which may or may not use F.E. methods at the detail level. If F.E. methods are used we give the process the special name **implicit mesh refinement.**

As far as the global analysis is concerned, these issues are transparent; there is absolutely no complication of the analysis compared with a straightforward model with no local features. Once more we are faced with a judgment which owes more to engineering appreciation of behaviour and practicalities of analysis operation than to finite element theory. In table 4 we list some of the influencing considerations.

TABLE 4

Factors Indicating Explicit or Implicit Modelling

For Explicit Modelling	For Implicit Modelling
Large scale of features (comparable with or greater than basic mesh size)	Small scale of individual features (relative to basic mesh)
Major fixtures such as mounting fittings carrying significant loads	Large numbers of similar features
Critical importance of investigating the features, per se, particularly if.....	Adequate theoretical treatment, given characteristic load levels (e.g. fracture mechanics, standard stress raisers, etc.)
No adequate local treatment available	

4.2 Explicit Modelling - Nodes or Know-how?

When we decide to represent features explicitly we are still faced with an important choice - whether to use a relatively coarse representation, relying on understanding and subsidiary analysis to fill in the detail or whether to "bury the problem in nodes" and let the F.E. method sort out the details. In the extreme, the latter approach is cited as a potential use of supercomputers - to solve complex structures without any need for analytical skill. In most cases, reliance on nodes in lieu of physical understanding is a dangerous practice. It is certainly expensive in solution time and possibly in data manipulation time too. There are, of course, cases where it is wholly appropriate.

For example in a very complex stress and deformation situation, such as might arise at an access panel in a laminated composite skin (with single sided reinforcement) as shown in fig. 4, local fine meshing may be the only feasible solution. The combination of edge effects, local fastener holes, asymmetric reinforcement and local non-linear behaviour makes it advisable to isolate the region as a substructure which can be analysed to a degree of refinement impractical over the structure as a whole.

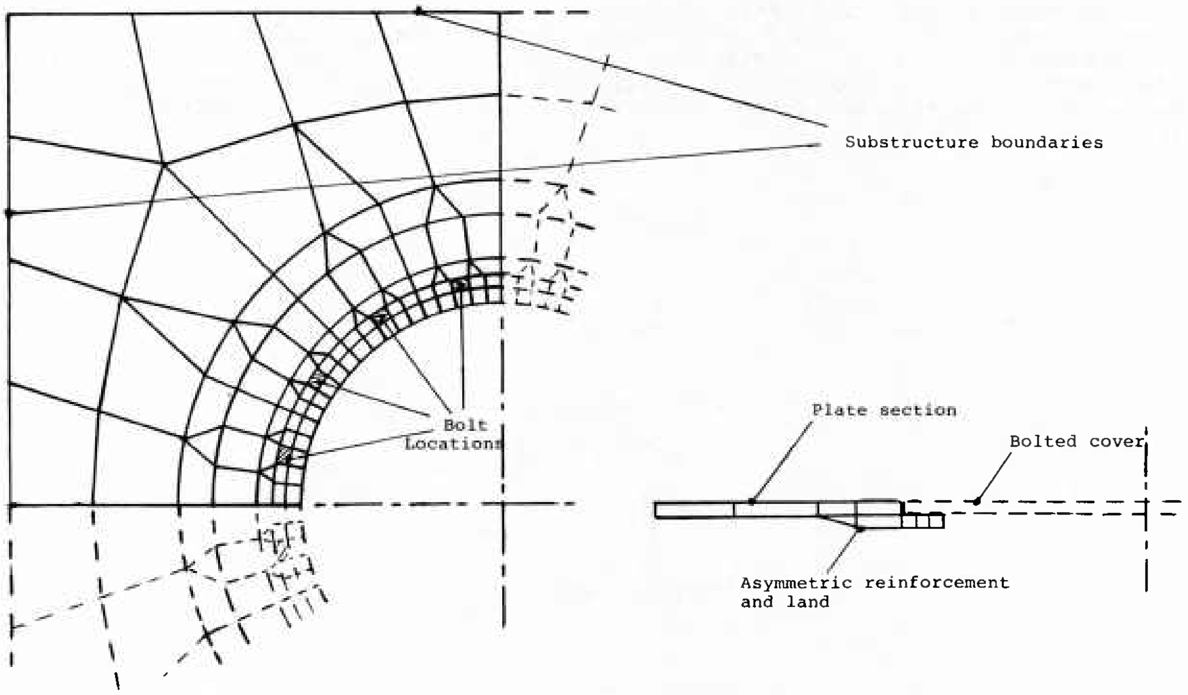


Fig 4 **Plate with Access Hole - Quadrant of Substructure**

More generally, however, we would recommend relatively coarse meshing, to obtain the stress or displacement trends in the region rather than directly estimating peaks. In the common case of stress raisers at cut-outs or fillets it is well known that the F.E. method usually displays the classic features of the law of dimimishing returns. In a NAFEMS benchmark test, (10, 11) typical convergence to a correct peak stress is illustrated in fig. 5

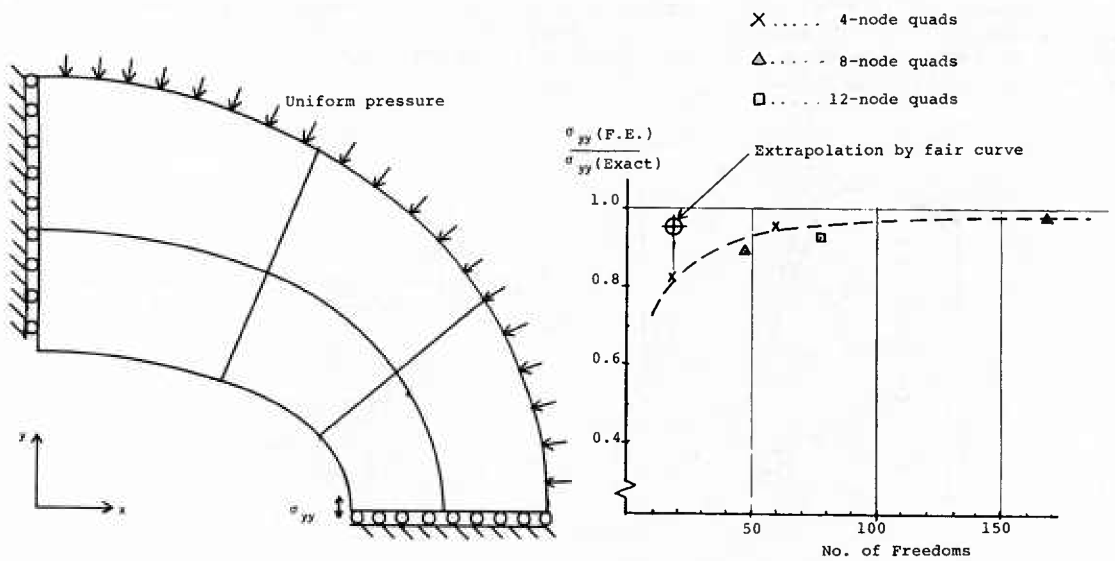


Fig. 5 **A NAFEMS Benchmark Test Showing Convergence Rate**

The coarsest solution, using a ring of 2 x 4-noded straight-sided quadrilaterals around periphery, returns 81% of the correct answer at the peak stress point. Using a faired curve either along the centre line or around the hole periphery the same analysis can yield 95% of the true solution, as good as we can obtain directly from three times as many nodes.

So we would suggest that, for modelling cut-outs and fillets, a normally fine mesh may use 3 elements, 4 or 7 nodes per quadrant around a curved boundary, 2 elements, 3 or 5 nodes per half side at a rectangle. At most, 2 rings of such elements may be needed before blending into the basic mesh. Very fine analysis may double the numbers of elements and beyond this we are dealing with individual cases to be treated on their merits.

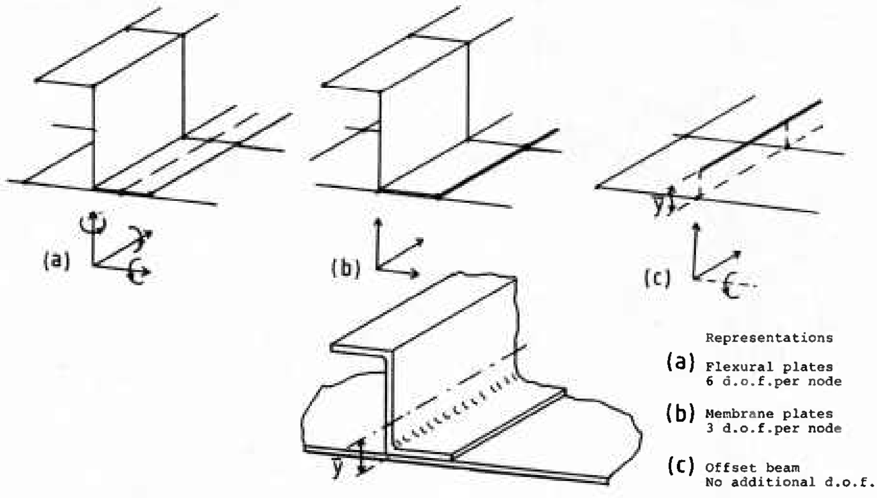


Fig. 6 Alternative Modelling of Skin Stiffener

Taking another quite different example, a stiffener supporting a skin, as shown in fig. 6, is better represented by a "cubic" beam with offset attachment and shear deformation (at 1 node x 4 d.o.f. per node, per longitudinal station) than by membrane flats (at 3 or 4 nodes x 3 d.o.f.). If we are concerned with interaction between stiffener and skin stability, we can either treat this by continuous stiffened skin analysis using stress levels derived from the beam representation or adopt a full flexural treatment of the stiffener allowing for attachment at the fastener line, needing 5 or 6 nodes x 6 d.o.f. per node. This complexity is very rarely justifiable as it is questionable whether there is any improvement in accuracy.

4.3 Explicit Modelling - Joints and Attachment Fittings

In airframe structure analysis, a most important, yet often neglected modelling consideration is the representation of joints and associated fittings. Bolted, riveted and even bonded joints can contribute significantly to overall structural flexibility. Attachment brackets, with their pins, bushes and fasteners, have a major influence on the distribution of load in our structures. In assessing the flexibility of joints it is always necessary to look at the details of load transfer because we often transmit substantial loads through thin material in flexure or via pins with significant offsets. However it is rarely necessary to represent this detail explicitly in any other than the most detailed local analyses.

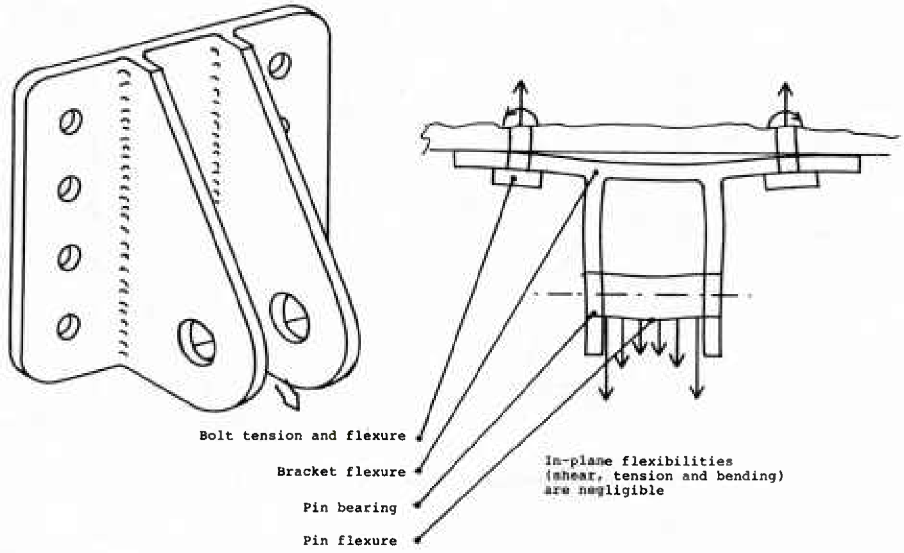


Fig. 7 Major Flexibility Contributions in a Mounting Bracket

Usually, explcit modelling of a joint and its fixtures means representing all major attachment points (e.g. lug centres) as structural nodes, either individually or at stations representing local node groups. It is always necessary to identify separate nodes on two sides of a joint and represent flexible material between them. In representing attachment fittings we need to be careful to provide stiff load paths in the plane of webs and flanges and flexible paths out of plane. Where significant loads are transmitted in flexure, as in the base of the bracket in fig. 7 (or the cleat in fig. 1 of the previous paper) it is important to remember that it is **flexibility**, not stiffness, which matters. Thus if we are to ignore anything in these fittings it is the high-stiffness in-plane-loaded parts, not the flexible, flexurally loaded parts of the load path. It is always better to make a simple engineering estimate of bracket flexibility by assuming load paths and summing strain energies in the varies members than to ignore the fittings for lack of meaningful data. Better still, use the implicit modelling approach of ref. 9 and run some typical F.E. analyses on various brackets and derive flexibilities at their attachment points therefrom. One or two analyses may be sufficient to define representative flexibilities for 10 or more flap/slat/aileron/airbrake mounting brackets or 100 cleats.

Simple lap, angle and buttstrap joints depend more on the flexibility of fasteners, bending in sheets and plates with offset loadings, than on the flexibility of the fittings. Until recently, data on joint flexibility have not been widely available, but this is now being rectified to some extent by ESDU(12). Fig 8 shows a typical curve for stiffness of titanium bolts in aluminium alloy skins.

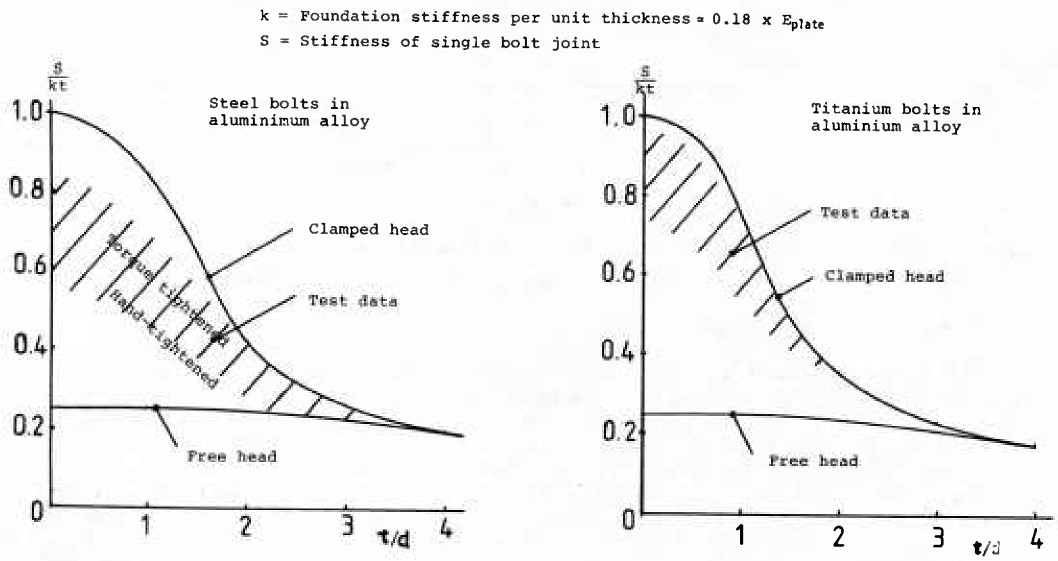


Fig. 8 **Bolted Joint Stiffness Data (Bolts in Single Shear)**

The commonest joints in sheet and plate structures are continuous line joints. In these it is rarely necessary to model individual fasteners, so data as in fig. 8 are normally used to estimate stiffnesses related to pairs of approximately coincident nodes at basic grid spacing. Stiffness at these nodes represents the aggregate behaviour of a group of fasteners and improved accuracy can be obtained by allowing, say, linear displacement variations between adjoining elements at a joint.

Individual fastener modelling, or at least higher order relative displacement functions, may be needed in load diffusion situations, such as the end of a stiffener or reinforcement. Fastener load peaks rapidly towards the end of the member so that the first two or three fasteners may carry the bulk of the load. Even in this case we can get very good results from a coarse analysis if we use a little subtlety in deriving the equivalent joint flexibilities. We can carry out a subsidiary calculation of load distribution at a stiffener end and calculate equivalent 'lumped' fastener flexibilities using, for example, a careful strain energy analysis. For complete load transfer at a multi-fasterner overlap joint, ESDU(11) provides a simple computer program to determine effective joint flexibility.

4.4 Implicit Modelling or Use of Equivalent Stiffnesses

As soon as we decide to perform an analysis at a scale larger than that of many structural features we must decide what to do about those features. Our options are:-

(i) model them explicitly and in some detail, as above

(ii) model them explicitly but at a coarse scale as suggested for continuous line joints.

(iii) represent them implicitly by modifying basic element properties

(iv) ignore them altogether and correct the results by subsequent local analysis

Option (iv) is favoured by most stressmen for obvious reasons and in many cases it is perfectly valid. The criterion to be used is whether or not the local features will have any discernible effect beyond the boundaries of the element in which they occur. Features which are continuous or frequently repeated, such as stiffeners on a skin or a series of lightening holes in a web, cannot and would not be ignored in any circumstances. Most sub-scale features can be classified as reinforcing (stiffeners, bosses, lands etc.) or weakening (holes, grooves, notches etc.) or joints. As a rough guide, such features can be ignored under the following circumstances.

Feature Type	Can be ignored if:-
Reinforcing features	Reinforcement is not continuous AND Aggregate volume of reinforcements <10% of basic element volume
Weakening features	No more than 20% section lost in any continuous loadpath across the element AND Aggregate volume of perforations <5% of basic element volume
Joints	Aggregate flexibility over periphery of element <10% of element flexibility under relevant uniform loading

These rough rules are intended to ensure that the strain energy in the element with its features is within $\pm$ 10% of the basic element under any relevant loading.

Coarse-scale explicit representation

This will be the normal method used for continuous reinforcing members, in rectangular or trapezoidal panels, as well as for many joints. The earliest way of representing stiffeners was by lumping into equivalent edge members, which is the crudest coarse-scale device. A better treatment in pure displacement or hybrid analysis is to apply consistent deformations along element boundaries using the identical interpolation forms used for sheet, plate or shell members and derive special stiffnesses at nodes, as in ref. 9. If not available as standard within the F.E. system, it is fairly simple to add one's own special elements on this basis. The alternative is a true implicit modelling method using "smearing" of stiffeners into an equivalent orthotropic sheet as discussed below.

Modification of basic element properties

This is usually the simplest method to introduce into a standard F.E. analysis as it involves no special mesh, no special elements and no solution complications. On the other hand it may require the most work by the analyst if it is to be used effectively. It is the recommended practice for dealing with most weakening features, discontinuous reinforcements and continuous reinforcements if special elements are too difficult to implement.

Considering membrane loading of plates as an example, then, if we assume that orthotropic material properties are available as a standard option, there are three material constants and an orientation angle which can be adjusted to simulate the behaviour of a modified basic element - four variables in all. In most cases it is sufficient to perform simple engineering analyses on single elements under constant direct shear and in-plane bending loadings in order to derive equivalent material properties. Many standard formulae(3, 11) are available to determine such stiffness equivalents for commonly occurring features. For example table 5 below is derived from ref. 3, in turn presented in ref. 11: it relates to the effective stiffness of shear webs with standard weakening features.

TABLE 5

Equivalent Stiffness of Weakened Shear Webs

Type of Web	Effective Shear Stiffeners (equivalent Gt)	Effective area A of web associated wih flange	
		Beams in flexure	Panels with low stress gradient
Plain webs	Gt	$\frac{bt}{6}$	$\frac{bt}{2}$
Honey comb sandwich webs	$\sum$ Gt for skins		
Shear-buckled plain web	0.6 Gt	Lesser of $15t^2$ or bt/6	$15t^2$
Web with lightening holes	$Gt\,(1-D/_a)$	$\frac{bt}{6}$	$\frac{(b-D)t}{2}$
Corrugated webs	$Gt\,(a/a_d)$	Zero normal to corrugations	Zero normal) to $\frac{bt}{2}$ parallel) corrugations
Web with shallow swages	Gt	Lesser of $15t^2$ or $\frac{bt}{6}$	$15t^2$ normal) to $\frac{bt}{4}$ parallel) swages
Castellated webs	$\dfrac{Gt}{1+\dfrac{\alpha\,\beta}{(1-\alpha)}+\dfrac{0.4\,\beta^3\gamma^2}{(1-\alpha)^3}}$	Zero along line of castellations	zero along) to line $\frac{bt}{3}$ normal) of) castellations

where:- a, b,t = web dimensions $\quad a_d$ = developed length $\quad$ D hole diameter

$\alpha = \dfrac{\text{notch width}}{\text{pitch}} \qquad \beta = \dfrac{2 \times \text{notch depth}}{\text{plate depth}} \qquad \gamma = \dfrac{\text{plate depth}}{\text{notch pitch}}$

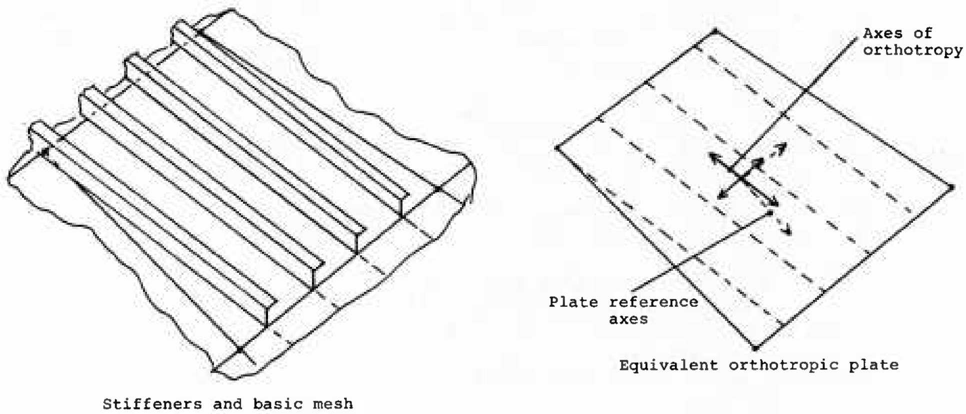


FIG. 9 **Modelling of Stiffeners Below Mesh Scale**

In many cases, such as the "smearing" of stiffeners into equivalent orthotropic sheet (fig. 9) the element modifications are self-evident; in this example, the effective longitudinal modulus is given by

$$E_{11}\,t \approx \frac{Et}{1-\nu^2} + \frac{E_s\,A_s}{b} \qquad \text{or } E_{11} \approx \frac{E}{1-\nu^2} + \frac{E_s A_s}{bt} \qquad \text{in the stiffener direction}$$

Where E is the skin Young's modulus and γ is Poisson's ratio
E_s is the stiffener modulus and A_s its area.

Another common application of element property modification arises when we wish to correct section dimensions to allow for varying distance from a neutral axis of bending(2). Typically we may use a $(\text{depth})^2$ correction for bending effectiveness of stiffeners which are idealised as though lying on a skin surface.

5. Specific Element Selection

Which elements we use to represent our structure depend, of course, on what we have available, and on our basic modelling strategy. In the case of plate and shell modelling there are usually most options available. We have already considered the membrane versus flexure argument and the low versus high order choice, but'it is worth re-stating some of our preferences:-

°Generally, quadrilateral elements out-perform triangles of comparable node density though triangles are often easier to fit into graded meshes; we always recommend quadrilaterals wherever possible, triangles only for mesh blending in non-critical areas.

°We prefer low order elements wherever we are sub-modelling (more features and details than mesh points) as they are simpler, easier to use and more reliable for the non-expert.

°We prefer higher order elements in continuum analysis and super modelling situations, for their better accuracy and economy in regions of stress concentration.

°We normally use pure displacement elements recognising that this is no more than historic attachment to familiar things.

We recognise the practical advantages of stress-based or hybrid elements in giving boundary representations which are closer to the stressman's requirements than displacement formulations but are wary of difficulties in recovering true displacements and of unpredictable performance outside proven applications.

°My own preference is for an element which does not yet exist in a satisfactory form, but which we hope to launch in the near future - a quadrilateral with in-plane nodal rotation (drilling) and a satisfactory capability to represent in-plane flexure without numerical integration fixes.

°In solid elements, the same basic rules apply: low order for sub-modelling, high order for supermodelling. There are some very bad performers about and it is worth while delving into element validation tests before making a final choice. Some useful special elements are available such as orthotropic shear elements to represent honeycomb core in sandwich structures.

6. <u>Load and Inertia Modelling</u>

In these days of automated aids to mesh generation (which is a purely geometric and topological problem), the preparation of loading data, which depend on element formulation as well as on factors wholly extraneous to the structural analysis, can become the dominant task in analysis data preparation. Two main factors can contribute to this problem:-

°Aerodynamic data are usually derived as load parameters related to a different mesh from the structure.

°Representation of continuous loading is only possible within the limited capabilities of the chosen structural elements.

The grid-to-grid transformation problem can prove quite tricky, especially in transonic and supersonic cases where abrupt loading changes are associated with shock fronts. Where point load values are used in both aerodynamics and structural representations, a direct, statically exact, transformation is desirable. Some general methods are available but more often the analyst must provide his own.

If pressure values are used in the structural analysis, these will normally be interpolated from the aerodynamic loads, leading to a possibility of small errors in static load balance (which may show up as more significant errors at supports or constraints). In any case a detailed load and moment balance check is an essential part of any such transformation. The process is very tedious and time consuming unless it is programmed in advance. We strongly recommend that all teams who regularly carry out structural analysis of flying surfaces should equip themselves with automated routines tailored to their requirements. This may appear to be a serious overhead cost, but the alternative is a great deal of repetitive, error prone and costly work for every analysis.

Whilst most structural analyses accept normal-to-surface loads either as pressure values or as equivalent nodal loads, depending on the complexity of the elements adopted, special attention is needed when distributed loads are applied in-plane at element boundaries. Here we must be careful to introduce kinematically equivalent loads(4) if we are to avoid serious distortion of stress patterns near to the boundaries. This again can be tedious and time consuming unless the job is handled via general purpose computer routines - so the same advice applies as before - equip yourself with the automation routine rather than repeatedly waste time and effort.

Inertia modelling is another difficult and time-consuming task if local accuracy is needed. In fine mesh analysis it is often adequate to use simplified distributed mass representations and many analysis systems provide a facility for specifying accelerations from which quasi-static inertia forces may be calculated. In coarse mesh analysis and most importantly in reduced fine mesh analysis (for dynamic response) it is rarely satisfactory to use simple lumping of masses at the coarse mesh nodes. We need to use kinematically equivalent inertia loads in direct coarse-mesh analysis and consistently transformed inertia matrices (which will include cross-coupling terms) when using reduced stiffnesses. Facilities provided in standard analysis packages are often inadequate to handle these jobs properly. In this paper we can do no more than recommend the analyst to seek the advice of reputable experts, if the system is inadequate.

7. Kinematic Boundary Conditions: Constraints and Symmetry

Many practical structural analysts, brought up on traditional, statics-based theory find difficulty in understanding kinematic boundary conditions and constraints. This difficulty is often compounded by the fact that the finite element systems handle commonly arising cases automatically, requiring only formalised data inputs from the user. Elegant treatments of symmetry as sub-cases of a universal concept, as in MSC/NASTRAN(13),can further distance the average engineer from real understanding. However, in the engineering theory of the previous lecture the concepts are simple, but as so often happens, applications of those simple concepts can become complex.

7.1 Constraints

Single point constraints, in which we usually equate one or more displacement components to zero to represent structural support, cause few problems. Multipoint constraints, which couple degrees of freedom at neighbouring points cause more difficulty. However, these are so commonly used that every analyst should make a conscious effort to understand them and acquire facility in describing and manipulating them. Some of the common applications are listed below:-

- applying symmetry conditions
- interface with engineers' theory analysis, e.g. "plane sections remain plane and undistorted"
- representing rigid members
- allowance for local offsets
- interpolation between otherwise incompatible meshes
- treatment of partial releases - hinges and slides
- introduction or elimination of special degrees of freedom (e.g. treatment of "drilling" rotation in shells)

Engineers' Theory Interface

This is a good example of the use of constraints because of its familiarity to every mechanical engineer. In fig. 10 we show an analysis mesh representing the end of a cantilever beam whose outer parts are adequately described by simple beam theory. At the interface we may either prescribe a set of forces as derived from standard beam theory or we impose the fundamental kinematic assumption: plane sections remain plane and undistorted, zero lengthwise displacement (pure flexure). In this simple two-dimensional example, this means defining the u-displacements as linearly varying with distance from the neutral axis and v- displacements as equal at all points.

There are always two way of writing down the constraint conditions:-

a) via a set of constraint equations expressing an imposed relationship between the nodal displacements

b) as a transformation relating the complete set of nodal displacements to a smaller number of displacement variables.

The second method is more general because the reduced variables need not be a subset of the nodal variables. They can be physically appropriate to the job. In this case, we would naturally choose vertical shear V and rotation θ as our variables. Mathematically the two formulations are equivalent and the constraint equations can always be derived from the transformation. In this case the two forms are as follows:-

Constraint equations

$$\begin{bmatrix} \frac{1}{y_1} & \frac{1}{y_2} & 0 & 0 \\ \frac{1}{y_1} & 0 & -\frac{1}{y_3} & 0 \\ \frac{1}{y_1} & 0 & 0 & \frac{1}{y_4} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = 0$$

$$\begin{bmatrix} 1 & -1 & 0 & 0 \\ 1 & 0 & -1 & 0 \\ 1 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} v_1 \\ v_2 \\ v_3 \\ v_4 \end{bmatrix} = 0$$

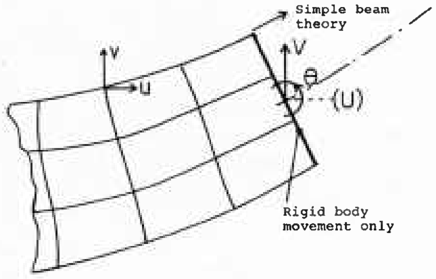


Fig. 10 Engineers' Bending Constraint

Transformation

$$\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ v_1 \\ v_2 \\ v_3 \\ v_4 \end{bmatrix} = \begin{bmatrix} 0 & y_1 \\ 0 & y_2 \\ 0 & y_3 \\ 0 & y_4 \\ 1 & 0 \\ 1 & 0 \\ 1 & 0 \\ 1 & 0 \end{bmatrix} \begin{bmatrix} V \\ \theta \end{bmatrix} \quad \text{or} \quad U = T \; \bar{U}$$

(8 displacements defined in terms of 2 remaining degrees of freedom)

The transformation version is in this case simpler, more physically meaningful and directly provides the means of representing the corresponding force relationship(4).

$$\bar{R} \equiv \begin{bmatrix} F \\ M \end{bmatrix} = T^T \{X_1 \; X_2 \; X_3 \; X_4 \; Y_1 \; Y_2 \; Y_3 \; Y_4\}$$

This is a special case of a **rigid member** constraint coupling four nodes. It allows total separation of the two parts of the beam with a consistent interface between them. It will most probably produce a stress distribution on the finite element side of the interface different from the EBT distribution on the other side, but static equivalence in terms of shear force and bending moment is assured.

Other Direct Constraint Conditions

Rigid elements, local offsets, local mesh interpolations can all be treated as above. In all cases the two ways of writing the constraints are available: in some cases one is the more natural in some cases the other. In all cases, transformation gives more generality and the added bonus of corresponding force definition. The transformation or constraint equations can always be written down by inspection from the geometry of the system. An important point to note is that all such geometric relationships must be **consistent** with nodal geometry to an order of accuracy **comparable with that used in the finite element progam,** otherwise significant errors in static equilibrium may arise. It is therefore recommended that computer routines be written (if not provided as standard) to derive all transformation coefficients directly from nodal geometry.

Partial Releases - Hinges and Slides

One standard treatment is to use duplicate nodes referenced by appropriate elements on either side of the release and to couple the constrained freedoms in just the same way as described above. The transformation route allows us to use relative displacement as one of the defining degrees of freedom rather than relate absolute displacements.

7.2 Symmetry and Repeated Boundary Conditions

It is common practice to analyse structures which have reflective or cyclic symmetry as single segments subject to appropriate loadings and kinematic constraints. In the extreme case, axial symmetry, we analyse a single cross section which is rotated around the axis of symmetry. If there are N repetitions of the basic segment (or **fundamental region**) then, in general, we can solve the structural problem completely by performing N analyses of the fundamental region. This means N different sets of boundary constraints and N corresponding sets of loading cases. For axisymmetric structures $N = \infty$ and we reduce the problem to a finite approximation by using Fourier series.

There are, of course, many practical situations where the principal loading cases of interest are themselves symmetric in some way: we may then reduce the number of separate solutions to the number of symmetric loading conditions.

Returning to the general case, it is useful to clarify our ideas about the treatment of symmetry by reference to a simple example - the singly-symmetric plate of fig. 11. A most important point to establish at the outset is that the analysis in terms of a repeated fundamental region is basically an application of super-position of loads and displacements: the various symmetric components (in this case the symmetric and anti-symmetric loads and constraints) apply to exactly the same structure. Provided that we retain the full force and displacement set at the axis of symmetry (including those freedoms set to zero in imposing the boundary conditions) the same stiffness matrix applies both to symmetric and antisymmetric systems. However, to relate the stiffness of a fundamental region to that of the structure as a whole clearly requires a transformation of displacements and forces. This transformation is needed for two purposes:-

i) to relate the co-ordinate systems on each side of a symmetry axis to a single global system

ii) to equate displacement at nodes on the symmetry axis for all repeated elements.

This transformation is **distinct from and must not be confused with** the displacement and force summation and decomposition relationships which connect symmetric sub-analyses with resultant solutions.

In our example, we analyse the right hand half of the symmetric plate in terms of symmetric displacements U_S and antisymmetric U_A (which include the zeros which prescribe the boundary conditions). The resultant displacements $\{U_r : U_l\}$ on right and left halves are given by:-

$$\begin{bmatrix} U_r \\ U_l \end{bmatrix} = \begin{bmatrix} I & I \\ I & -I \end{bmatrix} \begin{bmatrix} U_S \\ U_A \end{bmatrix}$$

on the assumption that U_l is defined in co-ordinates which are reflected about the axis, as shown in fig. 11. The inverse relationship gives the decomposition of resultant displacements into symmetric and antisymmetric parts:-

$$\begin{bmatrix} U_S \\ U_A \end{bmatrix} = \tfrac{1}{2} \begin{bmatrix} I & I \\ I & -I \end{bmatrix} \begin{bmatrix} U_r \\ U_l \end{bmatrix}, \text{ and for forces:-}$$

$$\begin{bmatrix} R_S \\ R_A \end{bmatrix} = \tfrac{1}{2} \begin{bmatrix} I & I \\ I & -I \end{bmatrix} \begin{bmatrix} R_r \\ R_l \end{bmatrix}$$

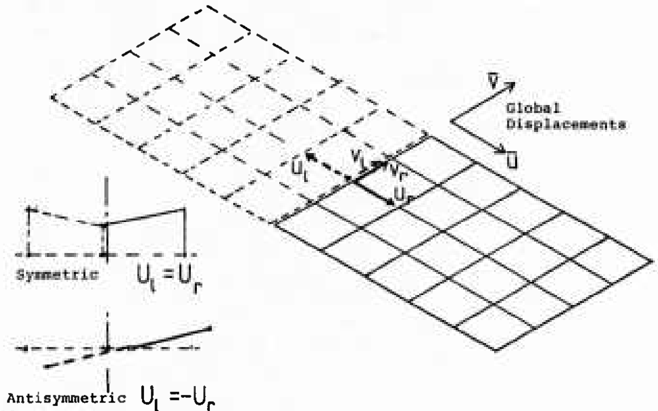


Fig. 11 Symmetric Plate

We re-emphasise that these are decomposition and super-position relations, **NOT** corresponding force and displacement transformations. [The symmetric and antisymmetric force **components** are ½ x the forces **corresponding with** symmetric and antisymmetric reflected **pairs** of displacements].

We associate the same stiffness K_0 for a half structure with both solutions, writing the stiffness equations as

$$R_S = K_0 \; U_S \quad \text{subject to } u_b = 0$$
$$R_A = K_0 \; U_A \quad \text{subject to } v_b = 0$$

where u_b and v_b are u- and v- displacements at the axis of symmetry. Different partitions of K_0 are, of course, used in the two solutions.

The quite distinct transformation which relates U_r and U_l to the global displacements $\bar{U}$ combines two functions as described above.

i) change the sign of U- displacements to the left of the axis

ii) equate $U'_{lb} = U'_{rb}$, the boundary displacements in left and right halves.

If U' represents the displacements in both separate structures referred to global axes then

$$U' = \begin{array}{c} \\ (U'_{rb}) \\ (U'_{lb}) \\ \\ \end{array} \begin{bmatrix} I & 0 & 0 \\ 0 & I & 0 \\ 0 & I & 0 \\ 0 & 0 & I \end{bmatrix} \begin{bmatrix} \bar{U}_r \\ \bar{U}_b \\ \bar{U}_l \end{bmatrix} = T_1 \, \bar{U}$$

Whilst, with different partitioning of U' we have

$$\begin{bmatrix} U_r \\ U_l \end{bmatrix} = \begin{array}{c} (u_r) \\ (v_r) \\ (u_l) \\ (v_l) \end{array} \begin{bmatrix} I & & & \\ & I & & \\ & & -I & \\ & & & I \end{bmatrix} U' = T_2 \; U'$$

The resulting transformation $T = T_1 \, T_2$ can be used in the normal way for expressing global stiffness in terms of the identical stiffnesses K_0 of the half structures

$$K' = T^T \begin{bmatrix} K_0 & 0 \\ 0 & K_0 \end{bmatrix} T$$

The same principles, involving the distinction between solution superposition and repeated sub-structure integration apply to all the more complex treatments of symmetry.

Conclusions

We have attempted, within the limited scope of a manageable lecture, to give a broad appreciation of practical issues in FE modelling, as seen from the viewpoint of an aerospace engineer concerned mostly with linear, quasi-static analysis. The presentation has touched upon many issues outside the realm of mesh generation which is the dominant subject in the extensive literature. None of these issues is treated here in depth, but we have tried throughout to draw attention to those basic principles which enable engineers to expand their understanding of a subject as they acquire experience in detail applications.

Most of the underlying ideas in this paper go back to the earliest days of finite element method development, when physical understanding was essential in order to use the very spartan and specialised tools then available. We observe today that there are many engineers who have slick facility in handling the "mechanics" of finite element analysis but lack that basic understanding which they bring to bear in traditional structural engineering. This is a gap which will not be closed by the current trend of burying structures in nodes. We are building up a vast capability for generating plausible nonsense faster and more convincingly than ever before.

We consider that particular attention needs to be given to recognising the important and unimportant features of a structure, a number of alternative ways of modelling those which are important and treatment of the tricky topics such as kinematic constraints. It is also considered important that we train engineers to use a simple 'language' of matrix structural analysis to provide a means of articulating basic concepts with precision but with minimum complexity. Finally, we see a growing role for expert systems to supplement the excellent manipulative facilities of modern computer analysis with simple, heuristic know-how. Combining the new ways with the old is a safer way forward than blind progression towards even bigger, more powerful black boxes.

References

1. NAFEMS Guidelines to Finite Element Practice. Department of Trade and Industry, National Engineering Laboratory, East Kilbride, Scotland 1984

2. Taig I.C. and Kerr, R.I. Some Problems in the Discrete Element Representation of Aircraft Structures in B. Fraeijs de Veubeke(ed.) AGARDograph 72, Pergamon 1964

3. Taig, I.C., Modelling and Interpretation of Results in Finite Element Structural Analysis presented at First World Congress on Finite Element Methods in Structural Mechanics, Bournemouth 1975

4. Taig, I.C., Stating the Problem: The Step Before F.E. Modelling in AGARD Lecture Series No. 147

5. Taig, I.C., Expert Aids to Reliable Use of Finite Element Analysis in Reliability of Methods for Engineering Analysis (D.R.J. Owen, ed.) Proceedings of Conference at Swansea 1986

6. Babuška, I., The Self-adaptive Approach in the Finite Element Method in J.R. Whiteman (ed) Mathematics of Finite Elements and Applications Academic Press, London 1975

7. Zienkiewicz, O.C., Gago, J.P. de S.R., Kelly, D.W., The Hierarchical Concept in Finite Element Analysis. Computers and Structures Vol.16 No. 1-4 pp 53-65 1983

8. Brandt, A., Multi-level Adaptive Solutions to Boundary Value Problems. Mathematics of Computing Vol. 31 No. 138 pp 333-390 1977

9. Taig, I.C., Implicit Modelling and Mesh Refinement for Complex Structures in International Journal for Numerical Methods in Engineering Vol. 22 pp 17-38 1986

10. Kamoulakos, A., Davies, G.A.O., Hitchings, D., Benchmark Tests for Various Finite Element Assemblies. Department of Trade and Industry, National Engineering Laboratory (Report C.1) East Kilbride, Scotland August 1985

11. Sharman, P.W. NAFEMS Membrane Stress Concentration Benchmark and the PAFEC Elements. Finite Element News Issue 3 pp 36-41 June 1986

12. ESDU Structures Sub-Series Vol 16 Component Idealisation for Finite Element Analysis. Data Items 84042, 85034, 85035 ESDU International Ltd. London

13. MSC/NASTRAN Hand book for Linear Static Analysis, R.H. MacNeal (ed) The MacNeal Schwendler Corporation, California U.S.A.

THE USE OF THE FINITE ELEMENT METHOD

V. B. Venkayya, Aerospace Engineer
Flight Dynamics Laboratory
Air Force Wright Aeronautical Laboratories
AFWAL/FIBRA, Wright-Patterson AFB OH 45433-6553

SUMMARY

These lecture notes are primarily intended to provide a quick overview of the solid mechanics problem for engineers using a general purpose finite element system in the solution of aerospace structures problems. It gives a brief outline of the solid mechanics problem and some of the available options for its solution. The finite element method is explained in more detail with particular emphasis on the use of membrane elements in aerospace structural analysis. The intention of these notes is to support a class room lecture.

1. INTRODUCTION

In the last thirty years the Finite Element Method (FEM) has developed into a powerful tool for solving a variety of engineering problems. These problems, at present, encompass a number of disciplines including aerospace, civil and mechanical engineering. The genesis of the FEM can be traced to the 1920's and 30's when civil engineers used it extensively for the static analysis of articulated frames in the name of slope deflections and moment distribution methods. These methods were well developed for mechanical calculators which were in vogue at the time. It the 1950's the emergence of the digital computer opened new vistas for numerical analysis in general and the finite element method in particular. Since then the FEM has grown rapidly from simple static strength analysis to extensive dynamic analysis of one, two and three dimensional structures problems. At the same time the scope of the method was extended to the solution of a variety of field problems including fluid mechanics, heat transfer, fluid-structure interaction, acoustic cavity analysis and a number of interdisciplinary problems. Now the method is no longer limited to linear analysis only. It has extensive applications in non-linear mechanics problems as well.

The decades of the 50's and 60's have seen intense research in element development, improvements to numerical solutions, and the associated sparse matrix manipulation schemes for the solution of large finite element assemblies. This was followed by the development of large scale applications software and innovative extensions in solid mechanics as well as other disciplines in the 1970's. In the 1980's there are at least 15 to 20 general purpose finite element programs being marketed throughout the world. These programs have extensive element libraries to meet the requirements of most complex engineering problems. At present the use of the FEM in mechanical design has become as common as the availability of electronic computers.

The extensive data preparation requirements of the FEM have spawned the development of user friendly pre and post processors which significantly increase productivity in the design office. They facilitate rapid error checking of the input data and interpretation of the output of large finite element models. However, availability of good finite element programs is not synonymous with their correct application. The pre and post processors do not necessarily assure a true correspondence of the mathematical model and the physical system. Proper modeling requires a thorough understanding of the physics of the problem as well as some understanding of the details of the theoretical basis of the program being used. Just a superficial understanding of the input instructions of a finite element system is inadequate, because it can lead to erroneous models which can give unconservative results and premature failures. It must be clearly understood that finite element programs are only sophisticated mathematical tools. Their use or abuse depends on the user's understanding of the problem and system limitations.

The finite element model of a physical system generally consists of a description of the geometry, material properties, boundary conditions and applied forces. The geometry involves the selection of an appropriate grid to represent the continuum, suitable elements to connect the grid, and the properties of the elements. The important decision to be made in selecting a grid is the spatial distribution of the mesh size. This distribution of the grid depends on the overall objectives of the analysis. If the purpose of the analysis is to determine the overall load paths of a large built-up structure, then a relatively coarse mesh is adequate and desirable from cost considerations. On the other hand, if the objective is to capture details of the steep stress gradients and discontinuities such as cracks and cutouts, then a finer mesh is required, at least around the stress raisers. It is advisable to handle these two objectives in separate models rather than in one big model. For example, in dynamic analysis where the interest is to determine the overall dynamic behavior as represented by the frequencies and mode shapes, a coarse model of a built-up structure is cost effective, while a detailed stress model would require a much finer grid. The basic tenet of discrete methods is that a finer mesh gives more accurate results. However, a finer mesh requires not only more computational effort but also is difficult to check for model errors. It is also worthwhile pointing out that accuracy improves with a finer mesh only when the elements capture the behavior of the structure reasonably well. Another observation to be made is that, in general, higher order elements require a coarser mesh and vice versa for the same accuracy.

In finite element modeling selection of appropriate elements is one of the most important decisions. Both the accuracy of the analysis and the cost are dependent on the type of elements used in the model. The behavior of the structural element can be described by one or more differential equations. These differential equations are in turn approximated by the so-called shape functions which are derived from basic polynomial functions. The more complex the behavior of the element, the higher is the order of the shape functions necessary to represent its behavior. For example, a simple truss element (a rod) transmits forces by uniform tension or compression. If the possibility of its buckling is excluded, its behavior can be represented by a first order differential equation or a linear polynomial function. The behavior of a three-dimensional beam, on the other hand, is more complex in the sense that the bending (about two axes), axial tension or compression, torsion and shear have to be represented by different differential equations. For example, bending and torsion representations require fourth order differential equations. Under certain symmetry conditions they can be uncoupled, otherwise they are usually coupled. To represent this behavior shape functions must be at least of the order of cubic polynomials. Similarly the lowest order polynomial representation of membrane plate behavior is linear, while higher order representations can only capture the stress gradients within the element itself. The bending behavior of plates is governed by a fourth order partial differential equation, and the corresponding polynomial approximations must be at least of the quintic order.

In making an analysis of large structural components such as a wing, fuselage etc., modeling with simple (low order) elements is most desirable. These simple models can provide reasonably accurate information about the overall load paths, and the simplicity of the elements allows easier interpretation of the results. They are also ideal for parametric studies in preliminary design and optimization. The use of higher order elements is appropriate while making a detailed analysis of local areas, such as a plate with a cutout or a crack or local buckling of a panel etc. In general, the higher order elements are less forgiving when there are violations of the basic assumptions from which they were derived. For example, lumping of masses or forces at grid points based on inspection or intuition is not acceptable in finite element models involving higher order elements. Consistent formulation is almost mandatory in such cases. Because of these limitations dynamic analysis of large built-up structures with significant non-structural mass attachments becomes quite cumbersome with higher order elements.

Modeling material properties of isotropic and/or anisotropic materials in the linear elastic range is relatively simple and presents no difficulties. When the materials behave nonlinearly or beyond the elastic range, modeling becomes more difficult because of the nonuniqueness of solutions associated with the loading and unloading sequences. Modeling of the boundary conditions is another very important issue in establishing the correspondence between the physical system and the mathematical model. The degree of supports (partial or full fixity) and the internal and external dependence of the motions of various degrees of freedom (single point or multipoint constraints) are some of the important considerations in developing boundary conditions of a finite element model. Correct representation of the boundary conditions is crucial for obtaining good results from an analysis.

The external or the internal environment of the system is described by the applied forces on the finite element model. These forces can be due to aerodynamic, thermal, gravity (body forces), centrifugal forces, etc. depending on the environment in which the system operates. Any errors in the force representation will be directly transmitted to the results of the analysis. Both lumped and consistent formulations can be used with reasonable accuracy in the case of linear elements. For higher order elements only consistent formulations are recommended. Similar rules apply for the mass representation in finite element models.

As pointed out earlier, the effective use of a general purpose finite element system requires a reasonable understanding of the formulation and the limitations of the system and an indepth understanding of the physical system and its behavior under the action of external forces. The next few sections provide a cursory background to solid mechanics problems in general and the finite element method in particular. Hopefully this background provides some guidelines for accurate modeling of practical structures.

2. SOLID MECHANICS PROBLEM

An aircraft structure is a deformable body, and the understanding of its behavior under the action of external and/or internal forces is essential for a successful design to meet the performance requirements. A typical aircraft operates in a severe dynamic environment. This dynamic environment is generally approximated by a set of equivalent static, dynamic, thermal and body forces for design purposes. These forces are assumed to be deterministic or treated as truly random in nature. The essential point is that we have the means to determine the loading conditions on a structure, so that its deformable behavior can be predicted and an adequate structure can be designed. The finite element method is most often used for predicting the behavior of structures subjected to loads. Even though the finite element method can be applied to the solution of a variety of engineering problems, its original development was in response to the solution of solid mechanics problems. A brief description of the extent and the scope of solid mechanics problems is appropriate before presenting the finite element method.

The deformable body shown in Fig. 1a is subjected to a set of continuous or discrete forces and boundary conditions. The body is assumed to be supported adequately to prevent any rigid body motion, so that its deformable behavior can be studied independently. For linear problems the rigid and deformable behavior can be determined independently, and the combined effect can be obtained by superposition.

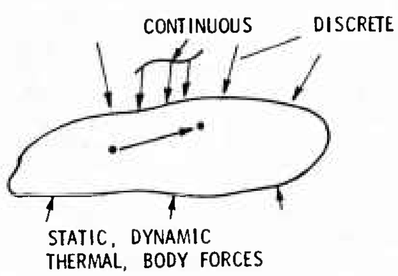


FIG. 1a. Deformable Body

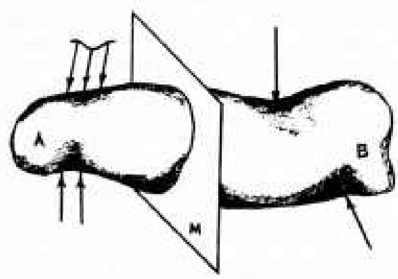


FIG. 1b. A cut through the Deformable Body

The state of the body after deformation can be defined by the displacement vector $\underset{\sim}{W}$, which represents the motion of a point from its initial position A to its new position $\underset{\sim}{A}'$. This displacement vector, $\underset{\sim}{W}$, can be defined by the three unknown displacement components u, v, and w respectively in the direction of the three orthogonal axes [1].

$$\underset{\sim}{W} = \begin{bmatrix} u(x, y, z, t) \\ v(x, y, z, t) \\ w(x, y, z, t) \end{bmatrix} \tag{1}$$

The three displacement components are functions of the spatial coordinates (x, y, z) of the point in the body. For dynamic problems they are also dependent on time. Knowledge of the displacements alone is not enough to determine whether the body can withstand the applied forces. For design we need to know the state of strains and stresses in the body. Fig. 1b shows a cut through the body to examine the state of the internal forces in the body. Fig. 1c shows the free-body diagram with internal stress resultants as force and moment vectors. Fig. 1d shows the normal and shear stress components on an infinitesimal element.

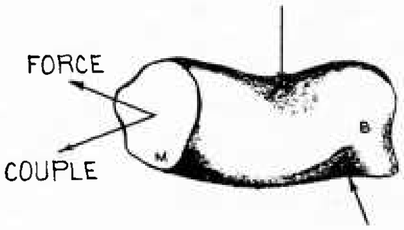


FIG. 1c. Free Body Diagram

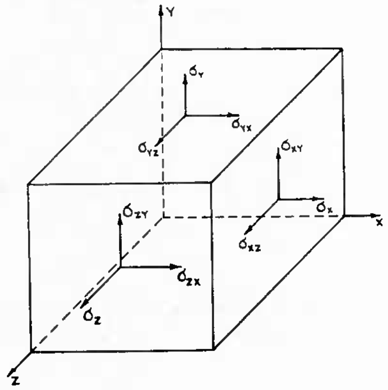


FIG. 1d. Stress Components on an Infinitesimal Element

The corresponding normal and shear strain components are implied. The six components of strain and the corresponding stresses are represented by

$$\underset{\sim}{\varepsilon}^t = [\varepsilon_{xx} \; \varepsilon_{yy} \; \varepsilon_{zz} \; \varepsilon_{xy} \; \varepsilon_{yz} \; \varepsilon_{zx}] \tag{2}$$

$$\underset{\sim}{\sigma}^t = [\sigma_{xx} \; \sigma_{yy} \; \sigma_{zz} \; \sigma_{xy} \; \sigma_{yz} \; \sigma_{zx}] \tag{3}$$

The solution of the solid mechanics problem implies a knowledge of these 15 unknowns as continuous functions of space and time. The list of 15 equations necessary to solve for the 15 unknowns is as follows:

Equilibrium equations	(3)
Strain-displacement relations	(6)
Stress-strain relations	(6)
Number of equations	(15)

The equilibrium equations derived from Newton's Laws can be written as follows:

$$\begin{aligned} \frac{\partial \sigma_{XX}}{\partial X} + \frac{\partial \sigma_{XY}}{\partial Y} + \frac{\partial \sigma_{XZ}}{\partial Z} + \bar{X} &= 0 \\ \frac{\partial \sigma_{YX}}{\partial X} + \frac{\partial \sigma_{YY}}{\partial Y} + \frac{\partial \sigma_{YZ}}{\partial Z} + \bar{Y} &= 0 \\ \frac{\partial \sigma_{ZX}}{\partial X} + \frac{\partial \sigma_{ZY}}{\partial Y} + \frac{\partial \sigma_{ZZ}}{\partial Z} + \bar{Z} &= 0 \end{aligned} \tag{4}$$

The equilibrium equations are written here in terms of the stress gradients and the body forces $\bar{X}$, $\bar{Y}$, $\bar{Z}$ on an infinitesimal element. When they are written in terms of stress resultants on a body with finite dimensions, the three equations translate into three force and three moment equations as follows:

$$\begin{aligned} &\sum_i F_{x_i} = 0 \qquad \sum_i F_{y_i} = 0 \qquad \sum_i F_{z_i} = 0 \\ &\sum_i M_{x_i} = 0 \qquad \sum_i M_{y_i} = 0 \qquad \sum_i M_{z_i} = 0 \end{aligned} \tag{5}$$

The six strain-displacement relations can be written as

$$\epsilon_{XX} = \frac{\partial u}{\partial X} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial X}\right)^2 + \left(\frac{\partial v}{\partial X}\right)^2 + \left(\frac{\partial w}{\partial X}\right)^2\right]$$

$$\epsilon_{YY} = \frac{\partial v}{\partial Y} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial Y}\right)^2 + \left(\frac{\partial v}{\partial Y}\right)^2 + \left(\frac{\partial w}{\partial Y}\right)^2\right]$$

$$\epsilon_{ZZ} = \frac{\partial w}{\partial Z} + \frac{1}{2}\left[\left(\frac{\partial u}{\partial Z}\right)^2 + \left(\frac{\partial v}{\partial Z}\right)^2 + \left(\frac{\partial w}{\partial Z}\right)^2\right]$$

$$\epsilon_{XY} = \frac{1}{2}\left[\frac{\partial v}{\partial X} + \frac{\partial u}{\partial Y} + \frac{\partial u}{\partial X}\frac{\partial u}{\partial Y} + \frac{\partial v}{\partial X}\frac{\partial v}{\partial Y} + \frac{\partial w}{\partial X}\frac{\partial w}{\partial Y}\right] \qquad (6)$$

$$\epsilon_{YZ} = \frac{1}{2}\left[\frac{\partial w}{\partial Y} + \frac{\partial v}{\partial Z} + \frac{\partial u}{\partial Y}\frac{\partial u}{\partial Z} + \frac{\partial v}{\partial Y}\frac{\partial v}{\partial Z} + \frac{\partial w}{\partial Y}\frac{\partial w}{\partial Z}\right]$$

$$\epsilon_{ZX} = \frac{1}{2}\left[\frac{\partial u}{\partial Z} + \frac{\partial w}{\partial X} + \frac{\partial u}{\partial Z}\frac{\partial u}{\partial X} + \frac{\partial v}{\partial Z}\frac{\partial v}{\partial X} + \frac{\partial w}{\partial Z}\frac{\partial w}{\partial X}\right]$$

The above strain-displacement relations contain both linear and non-linear terms. The linear strain-displacement relations can be written as

$$\epsilon_{XX} = \frac{\partial u}{\partial X} \qquad \epsilon_{XY} = \frac{\partial u}{\partial Y} + \frac{\partial v}{\partial X}$$

$$\epsilon_{YY} = \frac{\partial v}{\partial Y} \qquad \epsilon_{YZ} = \frac{\partial v}{\partial Z} + \frac{\partial w}{\partial Y} \qquad (7)$$

$$\epsilon_{ZZ} = \frac{\partial w}{\partial Z} \qquad \epsilon_{ZX} = \frac{\partial w}{\partial X} + \frac{\partial u}{\partial Z}$$

The same relations can be written symbolically in terms of a differential operator

$$\underset{\sim}{\varepsilon} = \underset{\sim}{D}\,\underset{\sim}{W}$$

where $\underset{\sim}{\varepsilon}$ and $\underset{\sim}{W}$ are the strain and displacement vectors and $\underset{\sim}{D}$ is the differential operator

$$\underset{\sim}{D}^t = \begin{bmatrix} \frac{\partial}{\partial X} & 0 & 0 & \frac{\partial}{\partial Y} & 0 & \frac{\partial}{\partial Z} \\ 0 & \frac{\partial}{\partial Y} & 0 & \frac{\partial}{\partial X} & \frac{\partial}{\partial Z} & 0 \\ 0 & 0 & \frac{\partial}{\partial Z} & 0 & \frac{\partial}{\partial Y} & \frac{\partial}{\partial X} \end{bmatrix} \qquad (8)$$

The six stress-strain relations are given by the generalized Hookes Law

$$\underset{\sim}{\sigma} = \underset{\sim}{E}\,\underset{\sim}{\varepsilon} \qquad (9)$$

where $\underset{\sim}{\sigma}$ and $\underset{\sim}{\varepsilon}$ are the stress and strain vectors and $\underset{\sim}{E}$ is the elastic constants matrix which represents the properties of the material. In an expanded form Eq. 9 can be written as

$$\begin{bmatrix} \sigma_{XX} \\ \sigma_{YY} \\ \sigma_{ZZ} \\ \sigma_{XY} \\ \sigma_{YZ} \\ \sigma_{ZX} \end{bmatrix} = \begin{bmatrix} E_{11} & E_{12} & E_{13} & E_{14} & E_{15} & E_{16} \\ & E_{22} & E_{23} & . & . & E_{26} \\ & & E_{33} & & & . \\ & & & E_{44} & & . \\ & \text{SYMMETRIC} & & & E_{55} & . \\ & & & & & E_{66} \end{bmatrix} \begin{bmatrix} \epsilon_{XX} \\ \epsilon_{YY} \\ \epsilon_{ZZ} \\ \epsilon_{XY} \\ \epsilon_{YZ} \\ \epsilon_{ZX} \end{bmatrix} \qquad (10)$$

The elastic constants matrix is symmetric about the diagonal. A general three-dimensional anisotropic body can, theoretically, have 21 independent elastic constants. Additional symmetry uncoupling reduces these to 9 elastic constants. An isotropic material has two elastic constants, the modulus of elasticity and Poisson's ratio or the two Lame's constants. A typical plane stress orthotropic material is characterized by four elastic constants. These are the moduli of elasticity in the longitudinal and transverse direction, the shear modulus and Poisson's ratio. Most fiber-reinforced composites are considered as plane-stress orthotropic materials.

The fifteen coupled differential equations presented in the foregoing discussion define the solid mechanics problem. However, additional relations must be considered to satisfy special requirements. For example, the six strain components are expressed as functions of only three independent displacements and as such there exists a dependency between the strains. This dependency is represented by the following compatibility conditions.

$$
\begin{aligned}
\frac{\partial^2 \epsilon_{XY}}{\partial X \partial Y} &= \frac{\partial^2 \epsilon_{YY}}{\partial X^2} + \frac{\partial^2 \epsilon_{XX}}{\partial Y^2} \\
\frac{\partial^2 \epsilon_{YZ}}{\partial Y \partial Z} &= \frac{\partial^2 \epsilon_{YY}}{\partial Z^2} + \frac{\partial^2 \epsilon_{ZZ}}{\partial Y^2} \\
\frac{\partial^2 \epsilon_{ZX}}{\partial Z \partial X} &= \frac{\partial^2 \epsilon_{ZZ}}{\partial X^2} + \frac{\partial^2 \epsilon_{XX}}{\partial Z^2} \\
2\frac{\partial^2 \epsilon_{XX}}{\partial Y \partial Z} &= \frac{\partial}{\partial X}\left(-\frac{\partial \epsilon_{YZ}}{\partial X} + \frac{\partial \epsilon_{ZX}}{\partial Y} + \frac{\partial \epsilon_{XY}}{\partial Z}\right) \\
2\frac{\partial^2 \epsilon_{YY}}{\partial Z \partial X} &= \frac{\partial}{\partial Y}\left(\frac{\partial \epsilon_{YZ}}{\partial X} - \frac{\partial \epsilon_{ZX}}{\partial Y} + \frac{\partial \epsilon_{XY}}{\partial Z}\right) \\
2\frac{\partial^2 \epsilon_{ZZ}}{\partial X \partial Y} &= \frac{\partial}{\partial Z}\left(\frac{\partial \epsilon_{YZ}}{\partial X} + \frac{\partial \epsilon_{ZX}}{\partial Y} - \frac{\partial \epsilon_{XY}}{\partial Z}\right)
\end{aligned}
\tag{11}
$$

Physically these compatibility conditions can be interpreted as assuring continuity in the deformation of the body (without breaks).

The solid mechanics problem must also satisfy the boundary conditions as given by

$$
\begin{aligned}
\sigma_{XX}\ell + \sigma_{XY}m + \sigma_{ZZ}n &= \bar{X} \\
\sigma_{YX}\ell + \sigma_{YY}m + \sigma_{YZ}n &= \bar{Y} \\
\sigma_{ZX}\ell + \sigma_{ZY}m + \sigma_{ZZ}n &= \bar{Z}
\end{aligned}
\tag{12}
$$

where $\bar{X}$, $\bar{Y}$, $\bar{Z}$ are the body force components in the x, y and z directions respectively, and ℓ, m, n are the direction cosines of the surface normal at the boundary.

The strain, stress and material property (elastic constants) transformations between the desired coordinate systems are the additional equations required in the solution of solid mechanics problems.

The strain transformation equation in three dimensions is given as

$$\underset{\sim}{\varepsilon}' = \underset{\sim}{T}_{\varepsilon}\, \underset{\sim}{\varepsilon} \tag{13}$$

where $\underset{\sim}{\varepsilon}$ and $\underset{\sim}{\varepsilon}'$ are the strains defined with respect to the x, y, z and x', y', z' axes respectively as shown in Fig. 2.

	X	Y	Z
X'	ℓ_1	m_1	n_1
Y'	ℓ_2	m_2	n_2
Z'	ℓ_3	m_3	n_3

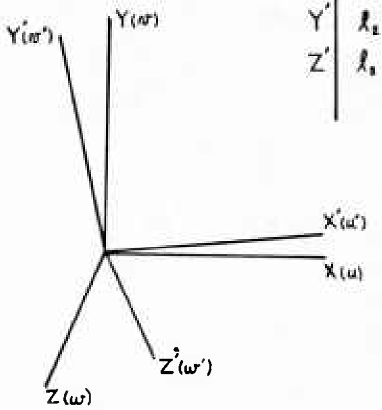


FIG. 2. Relationship Between the x, y, z Coordinate System and the x', y', z' Coordinate System

The strain transformation matrix is given by

$$
\underset{\sim}{T}_{\epsilon} = \left[\begin{array}{ccc|ccc}
\ell_1^2 & m_1^2 & n_1^2 & \ell_1 m_1 & m_1 n_1 & n_1 \ell_1 \\
\ell_2^2 & m_2^2 & n_2^2 & \ell_2 m_2 & m_2 n_2 & n_2 \ell_2 \\
\ell_3^2 & m_3^2 & n_3^2 & \ell_3 m_3 & m_3 n_3 & n_3 \ell_3 \\
\hline
2\ell_1 \ell_2 & 2m_1 m_2 & 2n_1 n_2 & (\ell_1 m_2 + \ell_2 m_1) & (m_1 n_2 + m_2 n_1) & (n_1 \ell_2 + n_2 \ell_1) \\
2\ell_2 \ell_3 & 2m_2 m_3 & 2n_2 n_3 & (\ell_2 m_3 + \ell_3 m_2) & (m_2 n_3 + m_3 n_2) & (n_2 \ell_3 + n_3 \ell_2) \\
2\ell_3 \ell_1 & 2m_3 m_1 & 2n_3 n_1 & (\ell_3 m_1 + \ell_1 m_3) & (m_3 n_1 + m_1 n_3) & (n_3 \ell_1 + n_1 \ell_3)
\end{array}\right]
\tag{14}
$$

If the strain transformation matrix is partitioned into 3x3 matrices, it can be written as

$$\underset{\sim}{T}_{\varepsilon} = \left[\begin{array}{c|c} T_{AA} & T_{AB} \\ \hline T_{BA} & T_{BB} \end{array}\right] \tag{15}$$

The submatrices $T_{AA},\ldots,T_{BB}$ can be identified from Eq. (14).

The stress transformation equations can be written as

$$\underset{\sim}{\sigma}' = \underset{\sim}{T}_{\sigma}\, \underset{\sim}{\sigma} \tag{16}$$

where the stress transformation matrix $\underset{\sim}{T}_{\sigma}$ is given in partitioned form as

$$T_{\sigma} = \left[\begin{array}{c|c} T_{AA} & 2T_{AB} \\ \hline \frac{1}{2}T_{BA} & T_{BB} \end{array}\right] \tag{17}$$

The elastic constants transformation matrix can be derived from the strain energy invariance condition as follows:

$$\underset{\sim}{E}' = \underset{\sim}{T}^{t}_{\varepsilon'}\, \underset{\sim}{E}\, \underset{\sim}{T}_{\varepsilon'} \tag{18}$$

where the strain transformation matrix $\underset{\sim}{T}_{\varepsilon}$ is given by

$$\underset{\sim}{T}_{\varepsilon'} = \underset{\sim}{T}^{-1}_{\varepsilon} = \underset{\sim}{T}^{t}_{\sigma} \tag{19}$$

The strain and stress transformation matrices for plane stress problems are given by

$$\underset{\sim}{T}_{\varepsilon} = \begin{bmatrix} \cos^2\theta & \sin^2\theta & \frac{1}{2}\sin 2\theta \\ \sin^2\theta & \cos^2\theta & -\frac{1}{2}\sin 2\theta \\ -\sin 2\theta & \sin 2\theta & \cos 2\theta \end{bmatrix} \tag{20}$$

$$\underset{\sim}{T}_{\sigma} = \begin{bmatrix} \cos^2\theta & \sin^2\theta & \sin 2\theta \\ \sin^2\theta & \cos^2\theta & -\sin 2\theta \\ -\frac{1}{2}\sin 2\theta & \frac{1}{2}\sin 2\theta & \cos 2\theta \end{bmatrix} \tag{21}$$

The fifteen coupled partial differential equations in fifteen unknowns can theoretically provide a means for the complete solution of three dimensional solid mechanics problems. The solutions must also satisfy the compatibility and boundary conditions. Among the fifteen unknowns the three displacement components are really the independent variables, and the remaining 12 unknowns can be expressed as functions of these three independent variables. Then we need to solve the three coupled partial differential equations which are given as

$$\begin{aligned} \nabla^2 u + \frac{1}{1-2\nu}\frac{\partial e}{\partial x} + \frac{\overline{X}}{G} &= 0 \\ \nabla^2 v + \frac{1}{1-2\nu}\frac{\partial e}{\partial y} + \frac{\overline{Y}}{G} &= 0 \\ \nabla^2 w + \frac{1}{1-2\nu}\frac{\partial e}{\partial z} + \frac{\overline{Z}}{G} &= 0 \end{aligned} \tag{22}$$

where ∇^2 is the three dimensional Laplacian operator

$$\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} + \frac{\partial^2}{\partial z^2} \tag{23}$$

and e is the volume dilatation given by

$$e = \varepsilon_{xx} + \varepsilon_{yy} + \varepsilon_{zz} \tag{24}$$

The solution of a general three dimensional solid mechanics problem by way of solving coupled partial differential equations, whether they are 15 or 3, is still an insurmountable task. We must find ways of simplifying the problem, even if it means limiting the scope of the problem. In the next section an outline is presented of some of the simplifications and methods available.

3. SOLUTION OF SOLID MECHANICS PROBLEM

As discussed in the previous section the general three dimensional solid mechanics problem consists of 15 unknowns and coupled partial differential equations for their solution. These solutions must also satisfy the strain compatibility and boundary conditions. In addition, the formulation and interpretation of the problem involves strain, stress, material properties and other transformations. In view of this complexity it is not realistic or possible to obtain a closed form solution for a general three-dimensional solid mechanics problem. A more realistic goal is to specialize these equations to specific problems whose behavior we can intuitively predict. Such specialization results in

* One dimensional problems
* Plane stress problems
* Plane strain problems
* Axisymmetric problems
* Bending and shear problems
* Inplane, bending, and shear problems
* Three-dimensional elasticity problems

An example of a one dimensional problem is a simple tension (compression) rod. Its behavior can be predicted by one first order differential equation. If this one dimensional element is subjected to bending (in one plane) in addition to the axial force, then its behavior can be predicted by two uncoupled differential equations, assuming that the axial force is small enough to neglect the coupling effects. A first order ordinary differential equation (ODE) predicts the axial deformation and a fourth order differential equation predicts the bending behavior. Similarly, if this line element is subjected to bending in a second plane and twisting about its own axis, then two more differential equations are necessary to predict their behavior. These four differential equations are coupled or uncoupled depending on whether the internal force coupling exists.

The plane stress and plane strain problems are two dimensional problems. Their behavior can be predicted by two first order partial differential equations. Similarly, axisymmetric problems can be described by a single ODE. The essential point of this discussion is that by limiting the scope of the problem based on the projected behavior, we can significantly reduce the complexity. However, this continuum approach based on the solution of differential equations imposes severe restrictions because of the continuity requirements and the need for satisfaction of the compatibility and boundary conditions. Because of these restrictions, the continuum (or the differential equations) approach is limited to simple components and loading conditions as shown in Fig. 3.

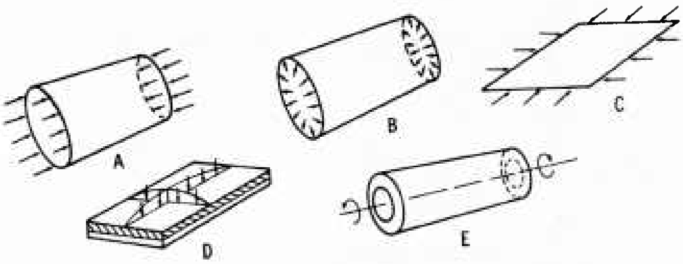


FIG. 3. Simple Components with Loading Conditions

A typical aircraft structure is built out of many structural components. It is inherently discontinuous and differential equation representation of the details is at best infeasible. For example, an aircraft wing shown in Fig. 4 consists of spars, spar caps, ribs, rib caps, skins and stiffeners. Spars, ribs and skins between the joints can be represented by plane stress or bending plate elements. Similarly, spar caps, rib caps and stiffeners between the joints can be represented by rods or beams. The behavior of each of these structural components is governed by different differential equations, and their behavior at the joints and across the joints is uncertain and cannot be adequately described by differential equations.

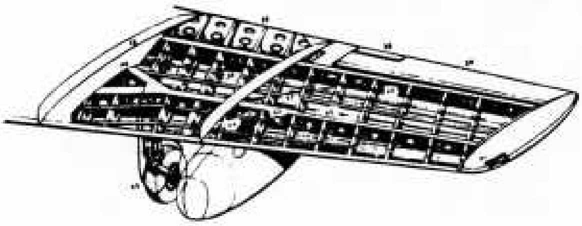


FIG. 4. Aircraft Wing

For such structures discrete approaches are more appropriate. The purpose of the discrete approach is to replace the governing differential equations by a set of algebraic equations whose solution can be adapted to a digital computer much more naturally. The basic principle behind the discretization is to obtain the solution of the problem at discrete points, instead of as continuous functions of the spatial coordinates. Then the solution between the discrete points can be obtained by interpolation or extrapolation. The first step in discretization is to transfer the effect of the continuum to preselected discrete points on the structure by the use of interpolation functions. This procedure is akin to the popular notion of lumping (low-level interpolation). Now the unknowns are the behavioral quantities (displacements) at the discrete points, and the relations are expressed in terms of algebraic equations. The second step is to solve for these unknowns at the discrete points. The third step is to obtain the solution between the points by the same interpolation as the first step.

Solid mechanics problems are generally classified into:

* Boundary value problems
* Initial value problems
* Mixed problems

A structure is a solid body of finite dimensions. The behavior of the body is prescribed at least partially at the boundary. The boundary conditions can be either kinematic (displacements, velocities, etc.) or in terms of forces. The name boundary value problem derives from this finiteness in spatial coordinates. Initial value problems refer, primarily, to the variable time. As in vibrations and heat conduction problems the initial state is prescribed but not necessarily at other times. For such problems if the state is prescribed at two different times, they will be called two point boundary value problems as distinct from the boundary value problems. A combination of initial and boundary value problems are called mixed problems. The vibration of beams, plates, etc. are some examples of mixed problems. The boundary refers to space, and initial refers to time in such problems.

There are a number of discrete methods for the solution of boundary value problems. Some of the prominent ones are:

* Finite differences
* Rayleigh-Ritz procedure
* Galerkin's Method
* Finite Element Methods
  * Stiffness Method
  * Flexibility Method

We will briefly outline the first three methods. The finite element method will be discussed in much more detail.

Initial value problems are generally solved by

* Collocation
* Subdivision
* Numerical integration
* Runge-Kutta Methods

Mixed problems are solved by

* Separation of the boundary and initial value problems.
* Combining both methods.

FINITE DIFFERENCE METHOD

The finite difference method can be summarized by the following six steps:

* Formulate the governing differential equations
* Approximate the structure by a discrete grid
* Apply the finite difference operator at each grid point
* Reduce the differential equation(s) to algebraic equations of the form

$$\underset{\sim}{K}\,\underset{\sim}{u} = \underset{\sim}{F}$$

* Solve the algebraic equations for the unknowns $\underset{\sim}{u}$
* Determine internal displacements, strains and stresses

Formulation of the governing differential equations in the first step can be accomplished either by equilibrium considerations or by a variational approach. Approximation of the continuum by a discrete grid involves selecting points in the structure where the behavioral variables like displacements, etc. can be determined by interpolation. For a given degree of interpolation the coarseness or fineness of the grid depends on the expected gradients of the behavior variables. High gradient regions need a finer grid and vice versa. The function of the difference operator is to replace the differential quantities by the unknown behavioral quantities at discrete points. An example of a central difference operator is given in

Fig. 5. The resulting set of algebraic equations contain a known coefficient matrix $\underset{\sim}{K}$, the unknown behavior variable (displacements) vector $\underset{\sim}{u}$ and a known vector $\underset{\sim}{F}$ (applied forces). The $\underset{\sim}{K}$ matrix is a function of the geometry, elements, and material properties of the structure. If the vector $\underset{\sim}{u}$ represents displacements, then the strains can be obtained by the strain-displacement relations which were given in the previous section. From the strains we can obtain the stresses using the stress-strain relations.

$$
\begin{array}{lcl|ccccc|l}
 & & & u_{i-2} & u_{i-1} & u_i & u_{i+1} & u_{i+2} & \\
\frac{\partial u}{\partial x} & = & \frac{1}{2h}\,[& & -1 & 0 & 1 & &] \\
\frac{\partial^2 u}{\partial x^2} & = & \frac{1}{h^2}\,[& & 1 & -2 & 1 & &] \\
\frac{\partial^3 u}{\partial x^3} & = & \frac{1}{2h^3}\,[& -1 & 2 & 0 & -2 & 1 &] \\
\frac{\partial^4 u}{\partial x^4} & = & \frac{1}{h^4}\,[& 1 & -4 & 6 & -4 & 1 &]
\end{array}
$$

FIG. 5. Central Difference Operator

RAYLEIGH-RITZ METHOD

As in the case of the finite difference method the Rayleigh-Ritz method can be summarized by the following steps:

* Formulate the total potential (strain energy and external work)

* Approximate the unknowns by a series

$$u(x, y, z) = \sum_{i=1}^{n} a_i \phi_i (x, y, z)$$

$\phi_i(x, y, z) \longrightarrow$ Coordinate functions - must satisfy the kinematic B.C.

$a_i \longrightarrow$ Unknown coefficients of the coordinate functions

* Minimization of the total potential w.r.t. the a^s gives a set of algebraic equations

$$\underset{\sim}{K}\,\underset{\sim}{a} = \underset{\sim}{F}$$

* Solve for the a^s and obtain the displacements from the second step.

* Determine the strains and stresses.

The Rayleigh-Ritz method gives a lower bound solution in the case of displacements and an upper bound solution in the case of frequencies etc. In other words the Rayleigh-Ritz approximation normally overestimates the stiffness of the structure.

GALERKIN'S METHOD

An outline of the Galerkin's method is as follows:

* Formulate the governing differential equations

* Approximate the solution either by a series or a polynomial

$$u(x,y,z) = \sum a_i \phi_i (x,y,z)$$

* Substitute the solution into the differential equations and obtain the error term e

* Make the weighted integral of the error over the region zero

$$\int \phi_i e dxdydz = 0$$

* Solve the resulting set of algebraic equations

$$\underset{\sim}{K}\underset{\sim}{a} = \underset{\sim}{F}$$

* Determine the strains and displacements

Galerkin's method is similar to the method of residuals frequently discussed in connection with the application of the finite element method to other engineering problems. The three methods discussed so far have some similarities and some differences. The last two methods give a symmetric system of equations. These methods are explained in the context of static analysis. However, their extension to dynamic analysis involves energies associated with the inertia and dissipation terms.

The last method we identified for the solution of boundary value problems is the finite element method. This method is also of primary interest to this lecture series. Before discussing this method in detail, we will introduce the familiar concepts of stiffness and flexibility and their implication in discrete structural analysis. In addition to providing an overview of the finite element method, the stiffness and flexibility discussion points out some similarities to the other three methods.

STIFFNESS AND FLEXIBILITY

We will introduce the concept of stiffness with the help of a simple linear spring (See Fig. 6) subjected to a force P at the end A which resulted in a displacement u at the same end. Since it is a linear spring, the displacement u will be linearly proportional to the applied load P, and this relation can be expressed by

$$P = K\,u \tag{25}$$

where K is the proportionality constant which represents the stiffness of the spring.

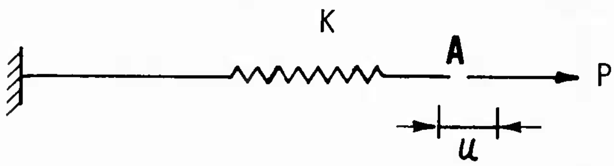


FIG. 6. Simple Linear Spring

From Eq. (25) the stiffness of the spring can be written as

$$K = \frac{P}{u} = \text{force per unit displacement}$$

So the stiffness of a structural element may be defined as the force necessary to produce a unit displacement. For example, if this spring is a simple tension/compression rod, the applied force P produces a uniform stress σ in the rod

$$\sigma = \frac{P}{A} \tag{26}$$

The strain in the rod is given by

$$\varepsilon = \frac{\sigma}{E} = \frac{P}{AE} \tag{27}$$

and the elongation u is given by

$$u = L\varepsilon = \frac{PL}{AE} \tag{28}$$

or

$$P = \frac{AE}{L}\,u \tag{29}$$

The axial stiffness of the rod is given by

$$K = \frac{AE}{L} \tag{30}$$

The flexibility of the spring, F, can be defined by

$$u = F\,P \tag{31}$$

and is given by

$$F = \frac{u}{P} = \text{displacement due to a unit force} \tag{32}$$

Then the flexibility and the stiffness of the spring are related by

$$F = K^{-1} \tag{33}$$

i.e. flexibility = the inverse of the stiffness and vice versa.

Now we will extend this concept of stiffness and flexibility to a built-up system consisting of many springs, displacements and force components. We will call this system a multi-degree of freedom system. The number of degrees of freedom of a system is defined by the number of independent coordinates necessary to fully identify its position in configuration space. For example, a particle in space can have a maximum of three degrees of freedom. These three degrees of freedom correspond to motion in the direction of the three orthogonal axes. However, the number of degrees of freedom of the particle can be less if its motion is restricted. For example, a particle restricted to plane motion will have only two degrees of freedom. Similarly, if it is limited to moving along a line, then it has only one degree of freedom. A rigid body in space can have a maximum of six degrees of freedom, three translational degrees of freedom corresponding to motion of the center of mass along the three orthogonal axes and three rotational degrees of freedom corresponding to rotations about the same axes. With the knowledge of these coordinates it is possible to completely define the position of the rigid body in space. As stated before, if any restrictions are imposed on the motion of the rigid body, then the number of degrees of freedom will be less than six. A deformable body in space will have an infinite number of degrees of freedom, since every point in the body can deform independently of the others, and there are an infinite number of points in the body when treated as a continuum. When this body is discretized by a finite number of points, then the degrees of freedom of the body will also be finite. Each of these discretized points can have up to a maximum of six degrees of freedom. So far the degrees of freedom of the system were defined with reference to the configuration space. Time dependence of the motion is not considered significant. However, if the motion is time dependent, then the state of the system cannot be defined by displacements alone. We must add to this degree of freedom definition velocities as well. Such a space is called a state space, and the number of degrees of freedom in the state space is twice that of the configuration space. The discussion in this lecture series is limited to the configuration space.

In Fig. 7a the point A is connected to three springs which in turn are fixed to supports at the other ends. Assuming that the point A has no finite dimensions, then it has two degrees of freedom if its motion is limited to the plane of the paper. The two degrees of freedom are identified by 1 and 2. u_1 and u_2 are the displacements, and P_1 and P_2 are the force components in the two directions respectively. The force-displacement relations for this two degree of freedom system can be written as

$$\underset{\sim}{P} = \underset{\sim}{K}\,\underset{\sim}{u} \tag{34}$$

where $\underset{\sim}{P}$, $\underset{\sim}{K}$ and $\underset{\sim}{u}$ are matrices and are identified by a wiggle under. The full matrix equation is written as

$$\begin{bmatrix} P_1 \\ P_2 \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \tag{35}$$

Now the first column of the stiffness matrix is defined as the force system necessary to maintain a displacement configuration in which $u_1=1$ and $u_2=0$. Fig. 7b shows the displacement configuration corresponding to the first column.

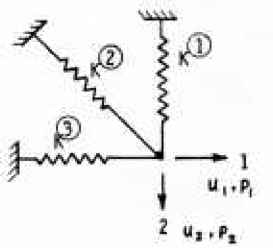


FIG. 7a. Two Degree of Freedom System

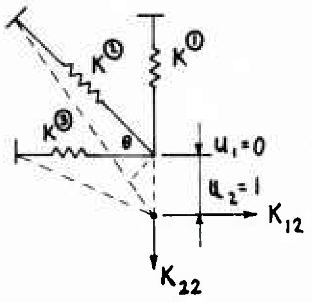


FIG. 7b. Displacement Configuration $u_1=1$, $u_2=0$

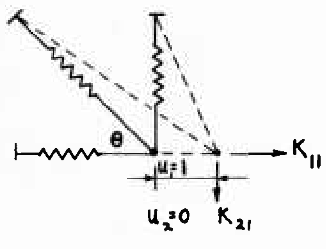


FIG. 7c. Displacement Configuration $u_1=0$, $u_2=1$

Similarly the second column of the stiffness matrix is defined as the force system necessary to maintain the displacement configuration in which $u_1=0$ and $u_2=1$. See Fig. 7c. The actual stiffness matrix is given by

$$\begin{bmatrix} P_1 \\ P_2 \end{bmatrix} = \begin{bmatrix} K^{(3)} + K^{(2)}\mathrm{Cos}^2\Theta & K^{(2)}\mathrm{Cos}\Theta\mathrm{Sin}\Theta \\ K^{(2)}\mathrm{Cos}\Theta\mathrm{Sin}\Theta & K^{(1)} + K^{(2)}\mathrm{Sin}^2\Theta \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} \tag{36}$$

So it is possible to construct the stiffness of a multi-degree of freedom system column by column by giving a unit displacement in the direction of each degree of freedom while all other degrees of freedom are fixed.

The flexibility of the same two degree of freedom system can be defined as

$$\underset{\sim}{u} = \underset{\sim}{F}\,\underset{\sim}{P}$$

$$\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_{11} & F_{12} \\ F_{21} & F_{22} \end{bmatrix} \begin{bmatrix} P_1 \\ P_2 \end{bmatrix} \tag{37}$$

The first column of the flexibility matrix can be defined as the displacements resulting from the application of a force system in which P_1=1 and P_2=0. Similarly the second column represents the displacements resulting from P_1=0 and P_2=1.

In summary the stiffness and flexibility can be defined as:

* The jth column of the stiffness matrix is a force system necessary to mintain a displacement configuration in which u_j=1 while all other u^s are zero.

* The jth column of the flexibility matrix is a displacement configuration resulting from a force system P_j=1 while all other forces are zero.

The element K_{ij} is the force in the ith direction due to a unit displacement in the jth direction while all other displacements are zero.

The element F_{ij} is the displacement in the ith direction due to a unit force in the jth direction while all other forces are zero.

For a given structure with the same degrees of freedom the relationship between the flexibility and the stiffness is given by

$$F = K^{-1} \tag{38}$$

provided $\underset{\sim}{K}$ is a non-singular matrix.

Fig. 8 gives the stiffness and the flexibility definitions for a three degree of freedom cantilever beam

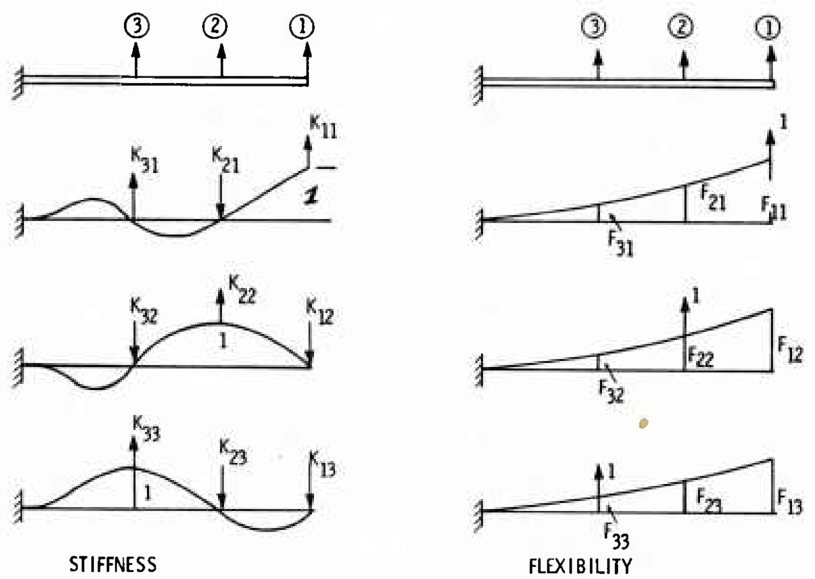


FIG. 8. Stiffness and Flexibility Definitions for a Three Degree of Freedom Cantilever Beam

In the stiffness method the displacements are the unknowns and the forces are the known quantities. By constructing the stiffness matrix using the properties of the structure, we can determine the unknown displacements. The reverse is true in the case of the flexibility method. The method outlined here for determining either the stiffness or the flexibility matrix is cumbersome to say the least when the number of degrees of freedom of the structure is large and the elements of the structure are more complex. In the next section a formal derivation of the displacement method of finite element analysis is presented. The displacement method is most amenable for implementation on a digital computer. The method is discussed in the context of plane stress (membrane) elements. Membrane elements are most suitable for determining the overall load paths of large built-up structures.

4. FINITE ELEMENT ANALYSIS

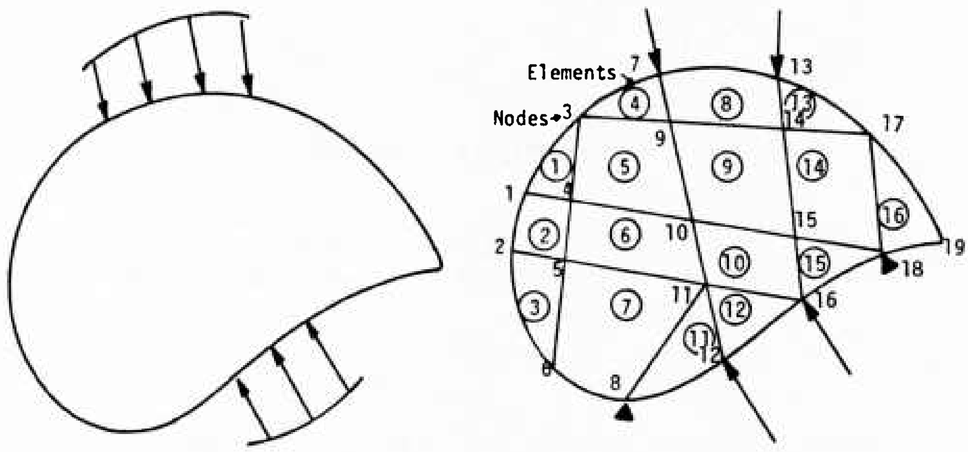
In the finite element analysis the continuum is replaced by a discrete model consisting of a finite number of nodes connected by elements (members) [2-5]. The rationale in such an approximation is that the response between the nodes (i.e., in the elements) can be expressed as a function of the response at the nodes. The functional relationship between the two responses is approximated by various interpolation functions or shape functions. The type of functions depends on the complexity of the problem at hand. This discretization reduces the original differential equations of the continuum to a set of algebraic equations which can be solved much more readily on digital computers.

The equations of the finite element analysis can be derived conveniently by considering the strain energy of the deformed system. For example, if the elastic body is idealized by m finite elements connecting q nodes (See Fig. 9), then the strain energy of the ith element can be written as

$$\tau_i = \frac{1}{2}\int_{V_i} \underset{\sim}{\sigma}_i^{t*}\ \underset{\sim}{\varepsilon}_i\ dV \tag{39}$$

where $\underset{\sim}{\sigma}_i$ and $\underset{\sim}{\varepsilon}_i$ are the stress and strain vectors and V_i is the volume of the element. For a linearly elastic body the relation between stress and strain can be written as

$$\underset{\sim}{\sigma}_i = \underset{\sim}{E}_i\ \underset{\sim}{\varepsilon}_i \tag{40}$$



(a) Continuum (b) Finite Element Model

Fig. 9. Continuum and Finite Element Model

where E_i is a symmetric matrix of material elastic constants. For typical plane stress problems the elastic constants matrix is of dimension 3x3. For an isotropic material in plane stress problems the elements of $\underset{\sim}{E}$ are as follows:

$$\underset{\sim}{E} = \frac{E}{1-\mu^2}\begin{bmatrix} 1 & \mu & 0 \\ \mu & 1 & 0 \\ 0 & 0 & \frac{1}{2}(1-\mu) \end{bmatrix} \tag{41}$$

where $\underset{\sim}{E}$ and μ are the elastic modulus and Poisson's ratio of the material respectively. For an orthotropic material the elastic constants matrix is given by

$$\underset{\sim}{E} = \frac{E_1}{1-\beta\mu^2}\begin{bmatrix} 1 & \mu\beta & 0 \\ \mu\beta & \beta & 0 \\ 0 & 0 & \frac{G}{E_1}(1-\beta\mu^2) \end{bmatrix} \tag{42}$$

where E_1 and E_2 are the longitudinal and transverse moduli, respectively, in the directions of the material property axes. β is the ratio of transverse to longitudinal modulus (E_2/E_1) and G and μ are the shear modulus and Poisson's ratio respectively.

The essence of the finite element approximation is that the internal displacements of the elements are expressed as functions of the displacements of the discrete nodes to which they are connected. As an example the local coordinate system and the nodal degrees of freedom of the triangular membrane element are shown in Fig. 10.

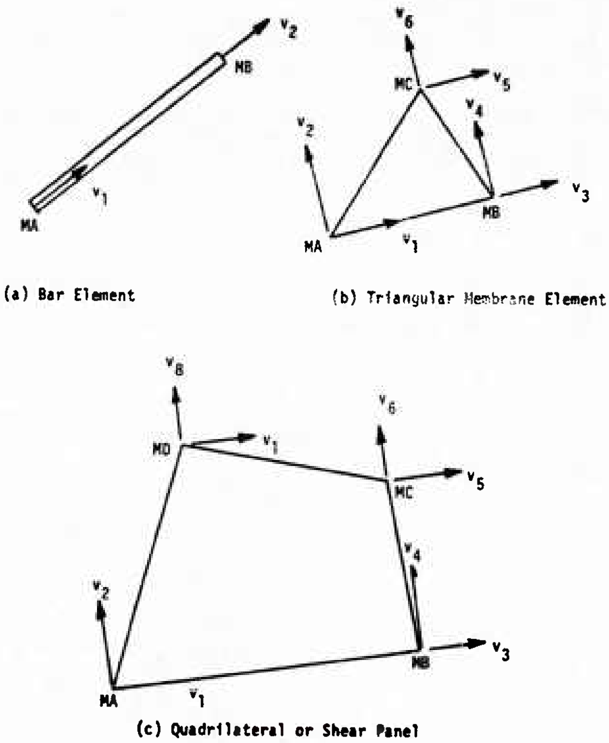


FIG. 10. Elements and Local Coordinate System

The functional relationship between the element's internal displacements and the discrete nodal displacements is given by

$$\underset{\sim}{w}_i = \underset{\sim}{\phi}_i \underset{\sim}{v}_i \tag{43}$$

where the matrix $\underset{\sim}{w}_i$ represents the displacements in the element which are functions of the spatial coordinates (x, y) for membrane elements. The shape function $\underset{\sim}{\phi}_i$ is a rectangular matrix, and its elements are also functions of the spatial coordinates. The vector $\underset{\sim}{v}_i$ represents the nodal displacements in the direction of the element degrees of freedom in the local coordinate system (Fig. 10). Now the strain-displacement relations can be written as

$$\underset{\sim}{\varepsilon}_i = \underset{\sim}{D} \underset{\sim}{w}_i \tag{44}$$

where $\underset{\sim}{D}$ is a differential operator. For a plane stress problem $\underset{\sim}{D}$ is given by

$$\underset{\sim}{D} = \begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{45}$$

Substitution of Equations 40, 43 and 44 in 39 gives the expression for strain energy in the following form

$$\tau_i = \frac{1}{2} \underset{\sim}{v}_i^t \underset{\sim}{k}_i \underset{\sim}{v}_i \tag{46}$$

where $\underset{\sim}{k}_i$ is the element (member) stiffness matrix with respect to the discrete coordinates $\underset{\sim}{v}$ and is given by

$$\underset{\sim}{k}_i = \int_{V_i} \underset{\sim}{\phi}_i^t \underset{\sim}{D}^t \underset{\sim}{E}_i \underset{\sim}{D} \underset{\sim}{\phi}_i \, dV \tag{47}$$

or

$$1 \times k_{pq} = \int_{V_i} \underset{\sim}{\sigma}_i^{(p)^t} \underset{\sim}{\varepsilon}_i^{(q)} \, dV \tag{48}$$

where $\sigma_i^{(p)}$ is the stress state due to the element displacement configuration in which $v_p=1$ while all other v's are zero. Similarly $\varepsilon_i^{(q)}$ is the strain state due to the unit displacement configuration in the direction of the qth degree of freedom. These two conditions are shown in Fig. 11 for the degrees of freedom 1 and 2 of the membrane triangle. It should be noted that besides assuming appropriate shape functions, the integration in Eqs. (47) or (48) is one of the difficult tasks in the case of complex elements in finite element analysis. However, for membrane elements this integration does not present any difficulties as will be seen in the next section. For more complex elements the usual practice is to adopt numerical integration schemes [6,7].

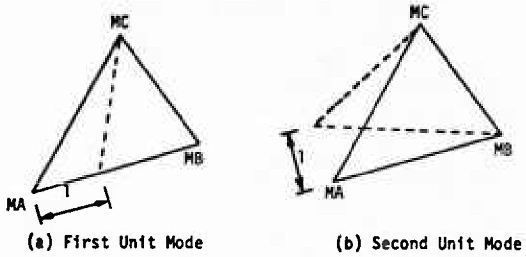


FIG. 11. Examples of Unit Displacement Modes

From Eq. (46) and Castigliano's first theorem, the relation between the element nodal forces and the displacements may be written as

$$\underset{\sim}{s}_i = \left[\frac{\partial \tau_i}{\partial v_j}\right] = \underset{\sim}{k}_i \, \underset{\sim}{v}_i \tag{49}$$

where $\underset{\sim}{s}_i$ is the element nodal force matrix corresponding to the displacement matrix $\underset{\sim}{v}_i$. Similar force-displacement relations for the total structure can be derived from the strain energy of the structure. The total strain energy Γ of the structure can be written as the sum of the energies of the individual components.

$$\Gamma = \sum_{i=1}^{m} \tau_i = \frac{1}{2} \sum_{i=1}^{m} \underset{\sim}{v}_i^t \, \underset{\sim}{k}_i \, \underset{\sim}{v}_i \tag{50}$$

In general, for most structures, it is convenient to define a local coordinate system for each element and a global coordinate system for the total structure. In such a case the element and structure generalized coordinates can be related by

$$\underset{\sim}{v}_i = \underset{\sim}{a}_i \, \underset{\sim}{u} \tag{51}$$

where $\underset{\sim}{a}_i$ is the compatibility matrix. Its elements can be determined by kinematic reasoning alone provided the structure is kinematically determinate. The matrix $\underset{\sim}{u}$ is the generalized displacement vector of the structure in the global coordinate system. It is interesting to note that Eq. (51) not only transforms element displacements from local to global coordinates but also gives information about how the elements are connected to the structure. From Eq. (51) and the principle of virtual work it is easy to show that the transformation between the forces on the structure and the element internal forces is given by

$$\underset{\sim}{P} = \underset{\sim}{a}_i^t \, \underset{\sim}{s}_i \tag{52}$$

where $\underset{\sim}{P}$ is the force vector on the structure in the global coordinate system. The transformation given in Eq. (52) is sometimes referred to as a contragradient transformation.

Substitution of Eq. (51) in (50) gives the expression for the total strain energy in the form

$$\Gamma = \frac{1}{2} \underset{\sim}{u}^t \, \underset{\sim}{K} \, \underset{\sim}{u} \tag{53}$$

where $\underset{\sim}{K}$, the total stiffness matrix of the structure, is written as the sum of the component stiffness matrices.

$$\underset{\sim}{K} = \sum_{i=1}^{m} \underset{\sim}{a}_i^t \, \underset{\sim}{k}_i \, \underset{\sim}{a}_i \tag{54}$$

Again using Castigliano's first theorem the relation between the generalized force matrix $\underset{\sim}{P}$ corresponding to the displacement matrix $\underset{\sim}{u}$ may be written as

$$\underset{\sim}{P} = \left[\frac{\partial \Gamma}{\partial u_j}\right] = \underset{\sim}{K} \, \underset{\sim}{u} \tag{55}$$

In most structural analysis problems the stiffness matrix $\underset{\sim}{K}$ is sparsely populated. It is essential to take advantage of this fact in solving the load deflection equations (Eq. 55), particularly in the case of problems with a large number of degrees of freedom where the cost of computation can be prohibitive otherwise. Gaussian elimination, with modifications to take into account the symmetry and sparseness of the stiffness matrix, is one of the effective methods for the solution of the load deflection equations.

Basically Gaussian elimination involves decomposition of the stiffness matrix by

$$\underset{\sim}{K} = \underset{\sim}{L}\ \underset{\sim}{D}\ \underset{\sim}{L}^{t} \tag{56}$$

where $\underset{\sim}{L}$ is the unit lower triangular matrix and $\underset{\sim}{D}$ is a diagonal matrix. The advantage of this decomposition scheme is that the $\underset{\sim}{L}$ matrix retains some of the sparseness characteristics of $\underset{\sim}{K}$ which consequently reduces the number of computations. Also $\underset{\sim}{L}$ and $\underset{\sim}{D}$ can be assigned the same storage as $\underset{\sim}{K}$.

The next step is the foward substitution by

$$\underset{\sim}{L}\ \underset{\sim}{Y} = \underset{\sim}{P} \tag{57}$$

where the matrix $\underset{\sim}{Y}$ is given by

$$\underset{\sim}{Y} = \underset{\sim}{D}\ \underset{\sim}{L}^{t}\ \underset{\sim}{u} \tag{58}$$

In Eq. (57) the solution of $\underset{\sim}{Y}$ can be accomplished by simple forward substitution. Once $\underset{\sim}{Y}$ is obtained, $\underset{\sim}{u}$ can be solved by back substitution using Eq. (58). The last two steps together are generally referred to as Forward-Back Substitution (FBS). Solution of Eq. (55) for multiple load vectors involves the decomposition of the stiffness matrix once and repetition of FBS as many times as there are load vectors.

With the help of these basic equations the steps in the finite element analysis can be outlined as follows:

1. Input information consists of

   a. Geometry of the structure

      Node Coordinates
      Element Connections
      Section Properties

   b. Material properties

   c. Boundary conditions

   d. Loading

   e. Clues for appropriate (desired) output.

2. Element information consists of

   a. Determination of the local coordinate system for each element.

   b. Selection of the appropriate shape functions (Eq. (43)).

   c. Determination of the element stiffness matrix (Eqs. (47) or (48)).

3. Transformation of the element stiffness matrix to the global coordinate system (Eq. (54) without summation).

4. Determination of the structure stiffness matrix by summation of the component stiffnesses (Summation in Eq. (54)).

5. Incorporation of the boundary conditions.

6. Solution of the load-deflection equations (Eqs. (55), (56), (57), and (58)).

7. Determination of the element displacements in their local coordinate system (Eq. (51)).

8. Determination of the stresses in each element (Eqs. (44), (43), and (40)).

9. Output the structure displacments, element stresses and other information such as element strain energies, etc.

The next section consists of the details of the stiffness matrix formulations for the four membrane elements and their application.

MEMBRANE ELEMENTS

The discussion in this section is limited basically to four finite elements:

1. Bar (Axial Force Member)
2. Membrane Triangle
3. Membrane Quadrilateral
4. Shear Panel

The four elements and their local coordinate systems are shown in Fig. 10. The bar is a constant strain line element and is equivalent to a rod element in the NASTRAN [8] program. The membrane triangle is a constant strain plate element similar to TRMEM in NASTRAN. The membrane quadrilateral is constructed out of four (non-overlapping) constant strain membrane triangles (element 2) with a fictitious interior node. This interior node is later removed by static condensation. This element is similar to QDMEM2 in NASTRAN. The shear panel is also constructed out of four non-overlapping triangles with a fictitious interior node. However, only the shear energy is considered in determining the stiffness of this element. Although the formulation is somewhat different, this element gives comparable results to the NASTRAN SHEAR element or the so-called Garvey shear panel [9].

The basis for the derivation of the shear panel is empirical, and it is primarily intended to elminate some of the difficulties encountered in using membrane triangles and quadrilaterals. For example, in beam problems (rectangular beams, I-beam, Box Beams including multicell wings and fuselage structures) the high stress gradients in the webs do not justify the use of constant strain triangles or quadrilaterals derived from these triangles. In fact, use of such elements for the webs (spars and ribs in wings) overestimates the stiffness significantly. Aerospace engineers have offset this difficulty to a large extent by judicious use of membrane elements in conjunction with the shear panels. In fact the early finite element models of wings and fuselages consisted primarily of bars and shear panels. However, the present practice of using membrane triangles and quadrilaterals for the top and bottom skins, bars for the posts, spar and rib caps, and shear panels for the spars and ribs eliminates to a large extent the need for determining the equivalent thicknesses and cross-sectional areas in the bar and shear panel model. The models consisting of these elements are most satisfactory for determining the primary load paths in built-up structures such as wings and fuselages. In addition the simplicity of these elements makes interpretation of the results easy and also keeps the analysis costs low because the stiffness matrices of these elements can be generated in a fraction of a second. Most of the remaining discussion of the membrane elements is extracted from References [10] and [11] which describe in detail programs ANALYZE and OPTSTAT.

BAR (ROD) ELEMENT

Basically this element is an axial force member. Its primary use is in two and three dimensional truss structures. It is also used extensively as spar and rib caps, posts around shear panels, stiffners and other line elements in aircraft structures. The local coordinate system of this element is shown in Fig. 10. The positive x-axis is directed along the line joining the two ends. v_1 and v_2 represent the element end displacements. The corresponding two end forces are s_1 and s_2. The displacement field in the element is assumed to be linear which gives constant strain. For a linarly elastic material this assumption yields constant stress as well.

If w, the displacement at any point along the length of the bar, is given by

$$w = ax + b \tag{59}$$

where a and b are two undetermined coefficients and x is the coordinate of the point in the local coordinate system , then the end displacements v_1 and v_2 are given by

$$\begin{bmatrix} v_1 \\ v_2 \end{bmatrix} = \begin{bmatrix} x_1 & 1 \\ x_2 & 1 \end{bmatrix} \begin{bmatrix} a \\ b \end{bmatrix} \tag{60}$$

where x_1 and x_2 are the coordinates of the two ends in the local coordinate system. Then the shape function (Eq. (43)) corresponding to this linear displacement field can be written as

$$\underline{\phi} = \frac{1}{(x_1 - x_2)} \left[(x - x_2), \; -(x - x_1)\right] \tag{61}$$

From the strain-displacement relations, the axial strain in the element is given by

$$\epsilon_x = \frac{\partial w}{\partial x} = a \tag{62}$$

From the principle of virtual work Eq. (48) the individual elements of the member sitffness matrix can be written as

$$k_{ij} = \int_V \sigma_x^{(i)} \, \epsilon_x^{(j)} \, dV = (-1)^{i+j} \frac{AE}{L} \tag{63}$$

where A is the cross-sectional area, L is the length of the member, and E is the modulus of elasticity of the material. The member stiffness matrix is given by

$$\underset{\sim}{k} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{64}$$

The member force matrix is given by

$$\underset{\sim}{s} = \underset{\sim}{k}\ \underset{\sim}{v} \tag{65}$$

The stress in the member is given by

$$\sigma_x = E\ \varepsilon_x \tag{66}$$

or

$$\sigma = \frac{s_1}{A} = \frac{-s_2}{A} \tag{67}$$

The strain energy in the element is given by

$$\tau_i = \frac{1}{2}\underset{\sim}{s}^t\ \underset{\sim}{v} \tag{68}$$

or

$$\tau_i = \frac{1}{2}\sigma_x\ \varepsilon_x\ A\ L \tag{69}$$

TRIANGULAR MEMBRANE ELEMENT

The membrane triangle is the basic plate element in the program. It is used to construct the membrane quadrilateral as well as the shear panel with some modifications. The membrane triangle can be used effectively in all cases where the primary loading is inplane forces. These include top and bottom skins of aircraft wings, flanges of I and box beams when they are subjected to constant normal stresses (tension or compression) only and skins of sandwich construction. Howewver, they are not suitable for situations where high stress gradients exist. For example, they are unsuitable for spars and ribs of wings and other lifting surfaces, webs of I and box beams and flat plates where the primary load is bending. If used in such cases, they overestimate the stiffness or generate singularity. Fig. 10 shows the triangular element with its local coordinate system. The generalized coordinates v_1, v_2, ---, v_6 represent the inplane displacements of the three nodes in the local coordinate system. The displacement field in the element is assumed to be linear. This gives constant strain in the element. For a linearly elastic material the stress in the element will also be constant.

The linear displacement field in the element can be represented by

$$\begin{aligned} w_x &= a_1\ x + b_1\ y + c_1 \\ w_y &= a_2\ x + b_2\ y + c_2 \end{aligned} \tag{70}$$

where w_x and w_y are the x-y displacements in the plane of the plate in the local coordinate system. a_1, b_1 etc. are the six undetermined coefficients. Eq. (70) can be written in matrix form as follows:

$$\underset{\sim}{w} = \begin{bmatrix} x & y & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & x & y & 1 \end{bmatrix} \begin{bmatrix} a_1 \\ b_1 \\ c_1 \\ a_2 \\ b_2 \\ c_2 \end{bmatrix} \tag{71}$$

The six unknown coefficients can be uniquely determined by the six boundary conditions at the nodes.

$$\begin{bmatrix} v_1 \\ v_3 \\ v_5 \\ \hline v_2 \\ v_4 \\ v_6 \end{bmatrix} = \left[\begin{array}{ccc|ccc} x_1 & y_1 & 1 & 0 & 0 & 0 \\ x_2 & y_2 & 1 & 0 & 0 & 0 \\ x_3 & y_3 & 1 & 0 & 0 & 0 \\ \hline 0 & 0 & 0 & x_1 & y_1 & 1 \\ 0 & 0 & 0 & x_2 & y_2 & 1 \\ 0 & 0 & 0 & x_3 & y_3 & 1 \end{array}\right] \begin{bmatrix} a_1 \\ b_1 \\ c_1 \\ \hline a_2 \\ b_2 \\ c_2 \end{bmatrix} \tag{72}$$

where x_1, y_1, ---, x_3 and y_3 are the coordinates of the three nodes of the triangle in the local coordinate system. It should be noted that the nodal displacements are grouped into x and y directions, so that the nodal coordinate matrix on the right hand side partitions into a diagonal matrix. The inversion of the partitioned diagonal matrix involves simply the inversion of the component matrix. Now the shape matrix $\underset{\sim}{\phi}$ is given by

$$\underset{\sim}{\phi} = \underset{\sim}{x}\, \underset{\sim}{Z}^{-1} \tag{73}$$

where the matrix $\underset{\sim}{x}$ is given by

$$\underset{\sim}{x} = \begin{bmatrix} x & y & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & x & y & 1 \end{bmatrix} \tag{74}$$

and the $\underset{\sim}{Z}$ matrix is given by

$$\underset{\sim}{Z} = \left[\begin{array}{c|c} \underset{\sim}{X} & 0 \\ \hline 0 & \underset{\sim}{X} \end{array}\right] \tag{75}$$

The coordinate matrix $\underset{\sim}{X}$ is given by

$$\underset{\sim}{X} = \begin{bmatrix} x_1 & y_1 & 1 \\ x_2 & y_2 & 1 \\ x_3 & y_3 & 1 \end{bmatrix} \tag{76}$$

It is interesting to note that each column of $\underset{\sim}{Z}^{-1}$ represents a unit displacement mode: i.e. the jth column of the inverse represents a displacement mode in which $v_j=1$ while all other nodal displacements are zero (See Fig. 11). This fact is used to advantage in determining the elements of the member stiffness matrix.

From linear strain-displacement relations the strains can be written as

$$\varepsilon_x = \frac{\partial w_x}{\partial x} = a_1 \tag{77}$$

$$\varepsilon_y = \frac{\partial w_y}{\partial y} = b_2 \tag{78}$$

$$\varepsilon_{xy} = \frac{\partial w_x}{\partial y} + \frac{\partial w_y}{\partial x} = b_1 + a_2 \tag{79}$$

From the principle of virtual work (Eq. (48)) the elements of the member stiffness matrix can be written as

$$k_{ij} = \int_V \underset{\sim}{\sigma}^{(i)^t}\, \underset{\sim}{\varepsilon}^{(j)}\, dV = \int_V \underset{\sim}{\varepsilon}^{(i)^t}\, \underset{\sim}{E}\, \underset{\sim}{\varepsilon}^{(j)}\, dV \tag{80}$$

where $\underset{\sim}{\sigma}^{(i)}$ and $\underset{\sim}{\varepsilon}^{(j)}$ are the stress and strain matrices corresponding to the unit displacement modes explained under Eq. (76). $\underset{\sim}{E}$ is the elastic constants matrix with respect to the element stiffness axis (See the local coordinate system of the triangular element in Fig. 10). If the material axis and the element stiffness axis coincide, $\underset{\sim}{E}$ would be the same as $\underset{\sim}{E}$ given in Eq. (42) for orthotropic materials.

In layered composite elements, however, the material axis and the element local axis do not generally coincide and transformation of $\underset{\sim}{E}$ to the element local axis is necessary before using it in Eq. (80). This transformation can be accomplished by considerations of energy invariance with axis rotation. For instance the element strain energy with respect to the material and the element local axes can be written as

$$\tau_m = \frac{1}{2}\, \underset{\sim}{\varepsilon}_m^t\, \underset{\sim}{E}_m\, \underset{\sim}{\varepsilon}_m \tag{81}$$

$$\tau_x = \frac{1}{2}\, \underset{\sim}{\varepsilon}^t\, \underset{\sim}{E}\, \underset{\sim}{\varepsilon} \tag{82}$$

where $\underset{\sim}{\varepsilon}_m$ is the strain matrix with reference to the material property axis. $\underset{\sim}{\varepsilon}$ is the strain matrix with reference to the element local axis. The strain matrices with reference to the material and element local axes are related by

$$\underset{\sim}{\varepsilon}_m = \underset{\sim}{T}\, \underset{\sim}{\varepsilon} \tag{83}$$

where $\underset{\sim}{T}$, the strain transformation matrix, is given by

$$\underset{\sim}{T} = \begin{bmatrix} \mathrm{Cos}^2\theta & \mathrm{Sin}^2\theta & \frac{1}{2}\mathrm{Sin}2\theta \\ \mathrm{Sin}^2\theta & \mathrm{Cos}^2\theta & -\frac{1}{2}\mathrm{Sin}2\theta \\ -\mathrm{Sin}2\theta & \mathrm{Sin}2\theta & \mathrm{Cos}2\theta \end{bmatrix} \tag{84}$$

and where θ is the angle between the element local axis and the material axis. By substituting Eq. (83) in (81) and invoking the condition of energy invariance with axis rotation, the expression for the elastic constants transformation can be written as

$$\underset{\sim}{\varepsilon} = \underset{\sim}{T}^t\, \underset{\sim}{E}_m\, \underset{\sim}{T} \tag{85}$$

The linear displacement variation in Eq. (70) implies constant strain, therefore the integral in Eq. (80) can be replaced by the volume of the element:

$$k_{ij} = \frac{1}{2}\, |\underset{\sim}{X}|\; t\; \underset{\sim}{\varepsilon}^{(i)^t}\, \underset{\sim}{E}\, \underset{\sim}{\varepsilon}^{(j)} \tag{86}$$

where $|\underset{\sim}{X}|$ is the determinant of the nodal coordinate matrix which represents twice the area of the element and t is the thickness of the element. Now the stiffness matrix of the element is given by

$$\underset{\sim}{k} = \frac{1}{2}\, |\underset{\sim}{X}|\; t \begin{bmatrix} \underset{\sim}{\varepsilon}^{(1)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(1)} & \underset{\sim}{\varepsilon}^{(1)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(2)} & \cdots & \underset{\sim}{\varepsilon}^{(1)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(6)} \\ \underset{\sim}{\varepsilon}^{(2)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(1)} & \underset{\sim}{\varepsilon}^{(2)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(2)} & \cdots & \underset{\sim}{\varepsilon}^{(2)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(6)} \\ \vdots & \vdots & & \vdots \\ \underset{\sim}{\varepsilon}^{(6)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(1)} & \underset{\sim}{\varepsilon}^{(6)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(2)} & \cdots & \underset{\sim}{\varepsilon}^{(6)^t}\underset{\sim}{E}\,\underset{\sim}{\varepsilon}^{(6)} \end{bmatrix} \tag{87}$$

The stress matrix in the element is given by

$$\underset{\sim}{\sigma} = \underset{\sim}{E}\, \underset{\sim}{\varepsilon} \tag{88}$$

The stresses obtained by Eq. (88) are with respect to the element local axis. It is often necessary to tranform these to the material property axis. This transformation can be obtained by

$$\underset{\sim}{\sigma}_m = \underset{\sim}{T}_s\, \underset{\sim}{\sigma} \tag{89}$$

where $\underset{\sim}{\sigma}_m$ is the stress matrix with respect to the material axis. The stress transformation matrix from the element local axes to the material axis is given by

$$\underset{\sim}{T}_s = \begin{bmatrix} \mathrm{Cos}^2\theta & \mathrm{Sin}^2\theta & \mathrm{Sin}2\theta \\ \mathrm{Sin}^2\theta & \mathrm{Cos}^2\theta & -\mathrm{Sin}2\theta \\ -\frac{1}{2}\mathrm{Sin}2\theta & \frac{1}{2}\mathrm{Sin}2\theta & \mathrm{Cos}2\theta \end{bmatrix} \tag{90}$$

The member force matrix is given by

$$\underset{\sim}{s} = \underset{\sim}{k}\,\underset{\sim}{v} \tag{91}$$

The strain energy in the element is given by

$$\tau_i = \frac{1}{4}\,|\underset{\sim}{X}|\;t\;\underset{\sim}{\sigma}^t\;\underset{\sim}{\varepsilon} \tag{92}$$

or

$$\tau_i = \frac{1}{2}\,\underset{\sim}{s}^t\;\underset{\sim}{v} \tag{93}$$

The next important step in the evaluation of the stress state in an element is the selection of a suitable failure criteria because of the combined stresses (σ_x, σ_y and σ_{xy}) in the plate elements. For isotropic materials the energy of distortion or the Von-Mises criterion is accepted as most satisfactory. The effective stress according to this criterion is given by

$$\sigma_{eff} = (\sigma_x^2 + \sigma_y^2 - \sigma_x\,\sigma_y + 3\sigma_{xy}^2)^{1/2} \tag{94}$$

When the allowable stresses are different in different directions, the effective stress ratio (ESR) according to the modified energy of distortion criterion can be obtained by

$$ESR = \left[\left(\frac{\sigma_x}{XX}\right)^2 + \left(\frac{\sigma_y}{YY}\right)^2 - \left(\frac{\sigma_x\sigma_y}{XXYY}\right) + \left(\frac{\sigma_{xy}}{ZZ}\right)^2\right]^{1/2} \tag{95}$$

where XX and YY are the tension or compression allowable in the x and y directions respectively, and ZZ is the shear allowable. Then the margin of safety (MS) is determined by

$$MS = \frac{1-ESR}{ESR} \tag{96}$$

The requirement of a positive margin of safety constitutes a stress constraint in optimization.

The failure criterion as given by Eq. (95) is adequate for isotropic as well as equivalent orthotropic structures. However, in the case of fiber reinforced layered composite materials, the question becomes much more complicated and there is little agreement on the type of criterion to be used. The fiber failure, matrix failure, delamination, and the effects of cut outs and bolt holes can trigger different failure modes. It is difficult, if not impossible, to combine all these effects into a single neat failure criterion as in metal structures. The present practice consists of a number of emperical criteria whose justification sometimes appears to be more emotional than rational. A review of some of these criteria is given in References [12,13]. The "OPTSTAT" program uses the failure criterion given by Eq. (95) for isotropic and equivalent orthotropic structures. For layered composite structures the fiber failure is used as a failure criterion. However, it is a relatively simple matter to modify this criterion to suit other requirements.

The composite element in "OPTSTAT" consists of stacked orthotropic membrane elements. Each orthotropic element (layer) in the stack represents the combined effect of all the fibers in one direction. The stiffness of the composite element is obtained by adding the stiffnesses of the component orthotropic elements representing all the fiber directions. This addition of the stiffnesses can be written as

$$\underset{\sim}{k} = \sum_{j=1}^{\ell} \underset{\sim}{k}_j \tag{97}$$

where $\underset{\sim}{k}_j$ represents the stiffness of all the fibers in one direction and ℓ represents the number of fiber directions in the composite element. The matrix $\underset{\sim}{k}_j$ for each direction of fibers is determined by Eq. (87). It is also assumed, for the summation in Eq. (97) to be valid, that the stiffness matrices $\underset{\sim}{k}_j$ in each composite element are determined with respect to the same set of reference axis such as the local element axis.

The composite element in "OPTSTAT" has at present a provision for four fiber orientations. These fiber orientations are 0°, 90°, and $\pm 45^\circ$. It is further assumed that the composite element is made of a balanced laminate. By adjusting the relative percentages of the fibers, the optimum directional properties of the laminate can be obtained. In assessing the failure of the laminate a weighted average of the effective stress ratios is considered instead of the failure of the individual fibers. This weighted average ESR is computed by

$$ESR = \alpha_0\,ESR_0 + \alpha_{90}\,ESR_{90} + \alpha_{45}\,ESR_{45} + \alpha_{-45}\,ESR_{-45} \tag{98}$$

where α_0, α_{90}, α_{45} and α_{-45} are the percentage of fibers in the 0°, 90° and $\pm 45^\circ$ respectively. Similarly ESR_0, ESR_{90}, ESR_{45} and ESR_{-45} are the effective stress ratios of the 0°, 90°, 45° and -45° layers.

QUADRILATERAL MEMBRANE ELEMENT

The quadrilateral element is most frequently used to represent membrane skins unless the corners etc. require the use of the triangular element. Fig. 10 shows the local coordinate system and the generalized coordinates (displacements) v_1 through v_8. The element is assumed to be a flat plate, and all nodes are assumed to lie on a plane connecting the first three nodes (1, 2, and 3). In effect the warping in the element is ignored. This approximation results in an overestimation of the stiffness of a truly warped quadrilateral element. In most cases the effect of the approximation is small, and it can be further reduced by reducing the mesh size of the model in the regions of high warping. However, if the warp is too large, the quadrilateral should be broken up into two or more triangles.

As mentioned earlier, the stiffness of the quadrilateral element is determined by breaking it into four component triangles as shown in Fig. 12. A fictitious node in the quadrilateral is located by averaging the coordinates of the four nodes as given by

$$x_5 = \frac{x_1 + x_2 + x_3 + x_4}{4} \tag{99}$$

$$y_5 = \frac{y_1 + y_2 + y_3 + y_4}{4} \tag{100}$$

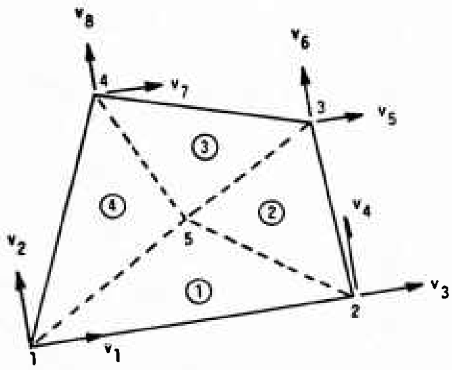


FIG. 12. Quadrilateral or Shear Panel Divided into Four Triangles

The stiffness of the four triangles is then computed by Eq. (87) in the local coordinate system shown in Fig. 10. Addition of the four stiffness matrices gives a 10 x 10 stiffness matrix with two degrees of freedom included for the fifth node. This fictitious node is later removed by static condensation before adding to the total structure. The procedure for static condensation is outlined next.

The force displacement relations of the 5 node quadrilateral are written as

$$\underset{\sim}{R}_Q = \underset{\sim}{k}_Q \, \underset{\sim}{r}_Q \tag{101}$$

where the subscript refers to the quadrilateral element with 5 nodes. Eq. (101), partitioned to isolate the degrees of freedom of the fifth node, can be written as

$$\left[\begin{array}{c} \underset{\sim}{R}_I \\ \hline \underset{\sim}{R}_{II} \end{array}\right] = \left[\begin{array}{c|c} \underset{\sim}{K}_{I,\,I} & \underset{\sim}{K}_{I,\,II} \\ \hline \underset{\sim}{K}_{II,\,I} & \underset{\sim}{K}_{II,\,II} \end{array}\right] \left[\begin{array}{c} \underset{\sim}{r}_I \\ \hline \underset{\sim}{r}_{II} \end{array}\right] \tag{102}$$

Eq. (102) can be written as two separate equations

$$\underset{\sim}{R}_I = \underset{\sim}{k}_{I,I} \, \underset{\sim}{r}_I + \underset{\sim}{k}_{I,II} \, \underset{\sim}{r}_{II} \tag{103}$$

$$\underset{\sim}{R}_{II} = \underset{\sim}{k}_{II,I} \, \underset{\sim}{r}_I + \underset{\sim}{k}_{II,II} \, \underset{\sim}{r}_{II} \tag{104}$$

Since the fifth node does not actually exist in the original model, no external forces can be applied to this node. This condition gives

$$\underset{\sim}{r}_{II} = -\underset{\sim}{k}^{-1}_{II,II} \, \underset{\sim}{k}_{II,I} \, \underset{\sim}{r}_I \tag{105}$$

Substitution of Eq. (105) in (103) gives

$$\underset{\sim}{R}_I = (\underset{\sim}{k}_{I,I} - \underset{\sim}{k}_{I,II}\, \underset{\sim}{k}_{II,II}^{-1}\, \underset{\sim}{k}_{II,I})\, \underset{\sim}{r}_I \tag{106}$$

From Eq. (106) the stiffness matrix of the original quadrilateral can be written as

$$\underset{\sim}{k} = \underset{\sim}{k}_{II} - \underset{\sim}{k}_{I,II}\, \underset{\sim}{k}_{II,II}^{-1}\, \underset{\sim}{k}_{II,I} \tag{107}$$

The stiffness as obtained by Eq. (107) is added to the total structure after appropriate coordinate transformations to the global coordinate system. When the structure displacements are determined, the fifth node displacements can be determined by Eq. (105). Now the stresses in each triangle can be determined as before. The effective stress ratio is determined for each triangle separately (Eq. (95)), and then a weighted average is used in computing the effective stress ratio and the margin of safety. This weighted average is computed by

$$\mathrm{ESR} = \frac{(\mathrm{ESR})_1\, \Delta_1 + (\mathrm{ESR})_2\, \Delta_2 + (\mathrm{ESR})_3\, \Delta_3 + (\mathrm{ESR})_4\, \Delta_4}{\Delta_1 + \Delta_2 + \Delta_3 + \Delta_4} \tag{108}$$

where $(\mathrm{ESR})_1$ through $(\mathrm{ESR})_4$ are the effective stress ratios of the four triangles. Δ_1 through Δ_4 are the respective planform areas of the triangles. In the case of fiber reinforced composite elements a further averaging across the thickness of the elements is used, as in Eq. (97), in determining the effective stress ratio. Now the margin of safety MS is computed as before by Eq. (96).

SHEAR PANEL

As the name indicates the shear panel is devised for the purpose of representing shear transmitting elements. For example, in wing structures the top and bottom skins can be represented by membrane (triangle and quadrilateral) elements. If the same elements are used for spars and ribs, the resulting finite element model grossly overestimates the stiffness of the structure. What this means is that the displacements obtained by this model will be smaller, or if this model is used for dynamic analysis, the frequencies of the structure will be much higher and cannot be matched with the results obtained from ground vibration tests. This behavior is due to the assumption of constant strain (stress) in the membrane element formulations. Most web elements in box or I-beams carry primarily shear and some normal stresses. In other words their deformation is primarily due to shear and not due to normal stresses. The normal stresses in webs usually have steep stress gradients, and the assumption of constant stress (or strain) is not justified. To offset this difficulty, and yet preserve the simplicity of the constant strain elements, a shear panel was formulated (Ref. 9) with the assumption that it carries only shear stresses. The bars and other membrane elements that surround the shear panel are supposed to carry the normal stresses. Such a situation does not actually exist in reality and thus the shear panel is an emperical element. However, the models built on such an assumption appear to produce satisfactory results.

Until recently it was a common practice in aircraft companies to model wings, fuselages, and empennage structures simply by bars and shear panels to obtain primary load path information. In such idealizations it was a common practice to assign a third of the cross-sectional area as spar and rib caps and the remainder for the shear panels. It should be pointed out that every shear panel must be surrounded on all four sides by normal stress carrying elements such as bars or membrane or bending elements. If the natural model does not contain such an element on any side of the shear panel, a nominal (or fictitious) bar (post) must be provided. Otherwise the model will have a singularity.

The shear panel can be constructed out of four triangles with the fictitious node inside as in the membrane quadrilateral discussed earlier. However, the stiffness matrices of the component triangles are determined by considering only the shear strain energy (Eq. (86)).

$$k_{ij} = \frac{1}{2}\, |\underset{\sim}{X}|\; t\; \varepsilon_{xy}^{(i)}\; G\; \varepsilon_{xy}^{(j)} \tag{109}$$

where G is the shear modulus, and $\varepsilon_{xy}^{(i)}$ and $\varepsilon_{xy}^{(j)}$ are the shear strains due to the unit displacement modes discussed earlier. There is one point that must be made here. The shear stress (strain) in an element changes with the orientation of the reference axis. Thus the stiffness matrix of the element can be sensitive to the reference axis. For rectangular elements the shear strain energy would be the same regardless of which side is selected for the reference axis. However, for quadrilaterals the stiffness matrix does depend on the reference axis. The errors produced by such departures are usually not significant, but it is worthwhile to make note of the assumptions involved.

As in the quadrilateral element the shear stresses in all four triangles are determined separately but with respect to the same reference axis. Of course, the normal stresses in the shear panels have no meaning. The margin of safety is determined by a weighted average of the effective stress ratios (ESR) as in the quadrilateral. The strain energy is determined by considering only the shear stress and strain. It should be noted that the shear panel can be used only as an isotropic or equivalent isotropic element.

There are three major steps in building finite element analysis software. The first step is deriving the appropriate shape functions for the elements, so that these elements can model the behavior of the structure adequately. The order of the shape function should reflect, at least to a degree, the order of the differential equation that models the physics of the problem. The volume integration indicated in Eq. (48) is the second major step. Numerical integration is most appropriate for handling arbitrary boundaries and higher order slope functions. Most finite element programs use Gaussian quadrature. The third major step is the solution of the load deflection equations. Gaussian elimination or variations of it are most appropriate for taking advantage of the symmetry and sparseness characteristics of the global matrices.

5. HIGHER ORDER INTERPOLATION AND SOLID ELEMENTS

So far we have discussed membrane elements with linear polynomial functions as displacement approximations. Here we will discuss the higher order interpolations, isoparametric formulations and the solid elements.

ROD ELEMENTS

Fig. 13 shows linear, quadratic and cubic polynomial approximations for displacements of the rod element.

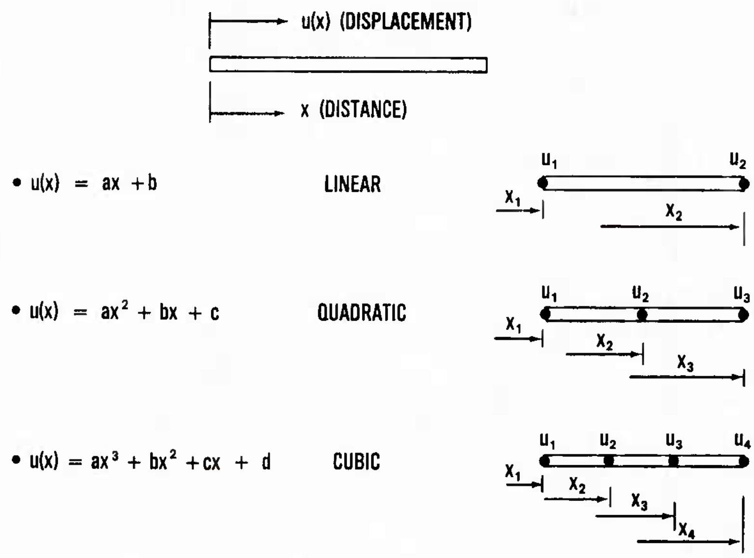


FIG. 13. Polynomial Approximations for Displacements of the Rod Elements

The number of grid points of the element are equal to the number of coefficients of the polynomial function. The functions for these elements are defined as

$$u = \sum_{i=1}^{m+1} \phi_i u_i \tag{110}$$

where m is the number of segments of the element, ϕ_i are the shape functions and u_i are the discrete displacements at the nodes or grid points. There are two ways to determine the shape functions. The direct way, as was done in the previous section, involves solving for the coefficients of the polynomial functions by assigning the discrete displacement values at the grid points whose coordinates x_i are known. For example the value of u_1 is given by

$$u_1 = ax_1^3 + bx_1^2 + cx_1 + d \tag{111}$$

for the cubic case.

Similarly by substituting the displacement values for the other grid points, the coefficients a, b, c and d can be solved in terms of the unknown displacements u_1, u_2, u_3 and u_4. Now sutstituting the coefficients into the original displacement equation, an expression for the shape function can be obtained. However, this procedure is cumbersome when the order of approximation is higher. The use of Lagrangian interpolation is much more appropriate for higher order approximations.

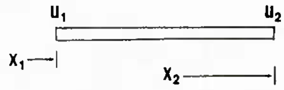
$$u = \sum_{i=1}^{m+1} \phi_i \, u_i \qquad m = \text{NUMBER OF SEGMENTS} \tag{112}$$

$$\phi_i = \frac{\prod\limits_{\substack{j=1\\ i\neq j}}^{m+1} (x - x_j)}{\prod\limits_{\substack{j=1\\ j\neq i}}^{m+1} (x_i - x_j)} \quad \text{LAGRANGIAN INTERPOLATION FUNCTION} \tag{113}$$

where ϕ_i is the ith shape function, x_i is the spatial coordinate of the ith grid point, and x is the distance at which the displacements are to be interpolated. The shape functions for each of the three approximations are given in Fig. 14.

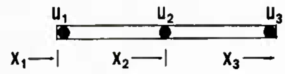
- **LINEAR** $u(x) = \phi_1 u_1 + \phi_2 u_2$

$$\phi_1(x) = \frac{x - x_2}{x_1 - x_2} \qquad \phi_2 = \frac{x - x_1}{x_2 - x_1}$$



- **QUADRATIC**

$$u(x) = \phi_1 u_1 + \phi_2 u_2 + \phi_3 u_3$$

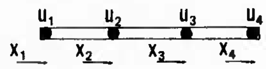


$$\phi_1(x) = \frac{(x-x_2)\ (x-x_3)}{(x_1-x_2)\ (x_1-x_3)} \qquad \phi_2(x) = \frac{(x-x_1)(x-x_3)}{(x_2-x_1)(x_2-x_3)}$$

$$\phi_3(x) = \frac{(x-x_1)(x-x_2)}{(x_3-x_1)\ (x_3-x_2)}$$

- **CUBIC**

$$u(x) = \phi_1 u_1 + \phi_2 u_2 + \phi_3 u_3 + \phi_4 u_4$$



$$\phi_1(x) = \frac{(x-x_2)(x-x_3)(x-x_4)}{(x_1-x_2)(x_1-x_3)(x_1-x_4)} \qquad \phi_2(x) = \frac{(x-x_1)(x-x_3)(x-x_4)}{(x_2-x_1)(x_2-x_3)(x_2-x_4)}$$

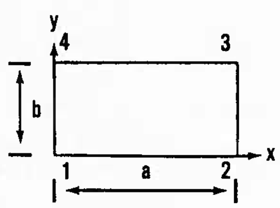
$$\phi_3(x) = \frac{(x-x_1)(x-x_2)(x-x_4)}{(x_3-x_1)(x_3-x_2)(x_3-x_4)} \qquad \phi_4(x) = \frac{(x-x_1)(x-x_2)(x-x_3)}{(x_4-x_1)(x_4-x_2)(x_4-x_3)}$$

FIG. 14. Shape Functions for the Rod Elements Using Lagrangian Interpolation

SHAPE FUNCTIONS - PLANE STRESS ELEMENTS

The shape function for a bilinear plane stress element is defined as before in Fig. 15.

BILINEAR ELEMENT



$$\underset{\sim}{u} = \underset{\sim}{\Phi}_e\, \underset{\sim}{u}_e$$

$$\underset{\sim}{u}^t = [u \ v]$$

$$\underset{\sim}{u}^t_e = [u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4]$$

$$\underset{\sim}{\Phi}_e = \begin{bmatrix} \Phi_1 & 0 & \Phi_2 & 0 & \Phi_3 & 0 & \Phi_4 & 0 \\ 0 & \Phi_1 & 0 & \Phi_2 & 0 & \Phi_3 & 0 & \Phi_4 \end{bmatrix}$$

$$u = \Phi_1 u_1 + \Phi_2 u_2 + \Phi_3 u_3 + \Phi_4 u_4$$

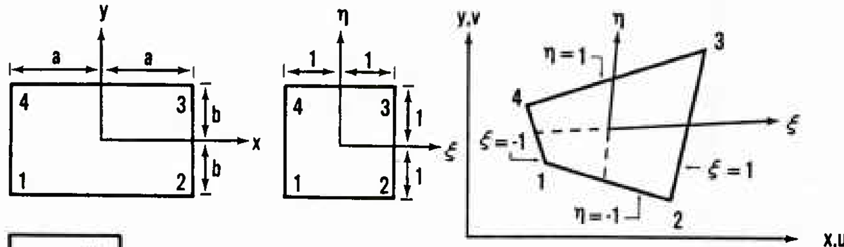
$$v = \Phi_1 v_1 + \Phi_2 v_2 + \Phi_3 v_3 + \Phi_4 v_4$$

$$\boxed{\begin{aligned} \Phi_1 &= \left(1-\tfrac{x}{a}\right)\left(1-\tfrac{y}{b}\right) \\ \Phi_2 &= \tfrac{x}{a}\left(1-\tfrac{y}{b}\right) \\ \Phi_3 &= \tfrac{x}{a}\tfrac{y}{b} \\ \Phi_4 &= \left(1-\tfrac{x}{a}\right)\tfrac{y}{b}. \end{aligned}}$$

FIG. 15. Shape Function for a Bilinear Plane Stress Element

This shape function represents linear displacements along the edges and a product of x and y terms in the interior. It represents an incomplete polynomial but nevertheless gives satisfactory results.

Extension of the bilinear formulation to an arbitrary quadrilateral can be facilitated by introducing the concepts of parametric mapping and isoparametric elements. When the displacements within an element and its boundary are approximated by the same shape functions, then the element is called an isoparametric element. This isoparametric mapping is shown in Fig. 16. Isoparametric mapping makes numerical integration over the domain of odd shaped elements easier, particularly in obtaining element stiffness matrices. Also in the case of higher order elements the curved boundaries can be modeled more accurately, because the boundary of the elements is also represented by higher order approximations.



$$\xi = \frac{x}{a} \qquad \eta = \frac{y}{b}$$

$$x = F_1(\xi,\eta)$$

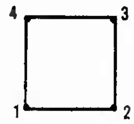
$$y = F_2(\xi,\eta)$$

$$\underset{\sim}{x} = \underset{\sim}{\phi}_e \, \underset{\sim}{x}_e$$

FIG. 16. Isoparametric Mapping

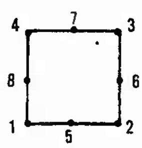
The new independent non-dimensional coordinates ξ and η are defined as shown. The shape functions for bilinear, biquadratic and cubic quadrilaterals are given in Fig. 17.

• BILINEAR ELEMENTS



$$\phi_i = \tfrac{1}{4}(1 + \xi\xi_i)(1 + \eta\eta_i)$$

• BIQUADRATIC ELEMENT



MIDPOINT NODES

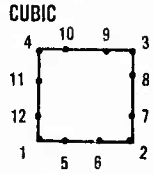
$\phi_i = \tfrac{1}{2}(1 + \eta\eta_i)(1 - \xi^2)$ ALONG $\xi_i = 0$

$\phi_i = \tfrac{1}{2}(1 + \xi\xi_i)(1 - \eta^2)$ ALONG $\eta_i = 0$

CORNER NODES

$\phi_i = -\tfrac{1}{4}(1 + \xi\xi_i)(1 + \eta\eta_i)(1 - \xi\xi_i - \eta\eta_i)$

• CUBIC



$\phi_i = 9/32\,(1 + \xi\xi_i)(1 - \eta^2)(1 + 9\eta\eta_i)$ AT $\xi_i = \pm 1$ $\eta_i = \pm \tfrac{1}{3}$

$\phi_i = 9/32\,(1 + \eta\eta_i)(1 - \xi^2)(1 + 9\xi\xi_i)$ AT $\eta_i = \pm 1$ $\xi_i = \pm \tfrac{1}{3}$

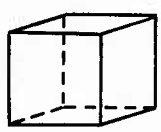
$\phi = 1/32\,(1 + \xi\xi_i)(1 + \eta\eta_i)\{9(\xi^2 + \eta^2) - 10\}$ AT CORNERS

FIG. 17. Shape Functions for Bilinear, Biquadratic and Cubic Quadrilaterals

The isoparametric formulation of the solid elements parallels that of the membrane elements. The shape functions for the linear quadratic elements are given by Fig. 18.

• LINEAR (8 NODES)

$$N_i = 1/8\,(1 + \xi\xi_i)(1 + \eta\eta_i)(1 + \zeta\zeta_i)$$



• QUADRATIC (20 NODES)

AT CORNERS

$$N_i = 1/8\,(1 + \xi\xi_i)(1 + \eta\eta_i)(1 + \zeta\zeta_i)(\zeta\zeta_i + \xi\xi_i + \eta\eta_i - 2)$$

AT $\xi_i = 0$ $\eta_i = \pm 1$ $\zeta_i = \pm 1$

$$N_i = \tfrac{1}{4}(1 - \xi^2)(1 + \eta\eta_i)(1 + \zeta\zeta_i)$$

AT $\eta_i = 0$ $\zeta_i = \pm 1$ $\xi_i = \pm 1$

$$N_i = \tfrac{1}{4}(1 - \eta^2)(1 + \xi\xi_i)(1 + \eta\eta_i)$$

AT $\zeta_i = 0$ $\xi_i = \pm 1$ $\eta_i = \pm 1$

$$N_i = \tfrac{1}{4}(1 - \zeta_i^2)(1 + \eta\eta_i)(1 + \xi\xi_i)$$

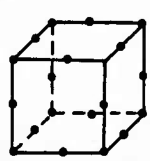


FIG. 18. Shape Functions for the Linear Quadratic Elements

The isoparametric mapping involves a change of coordinates from the x, y, z to the ξ, η, ζ system. This transformation must be reflected in the formulation of the element stiffness matrix. The element stiffness matrix in the x, y, z coordinate system and the required transformations to the ξ, η, ζ system are given in Fig. 19.

ELEMENT STIFFNESS EVALUATION

$$\underset{\sim}{k} = \int_V \underset{\sim}{\Phi}_e^t \, \underset{\sim}{D}^t \underset{\sim}{E} \, \underset{\sim}{D} \, \underset{\sim}{\Phi}_e \, dv$$

[J] ⟶ JACOBIAN MATRIX OF TRANSFORMATION

COORDINATE TRANSFORMATION

$$\frac{\partial \phi_i}{\partial \xi} = \frac{\partial \phi_i}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial \phi_i}{\partial y}\frac{\partial y}{\partial \xi} + \frac{\partial \phi_i}{\partial z}\frac{\partial z}{\partial \xi}$$

OR

$$\begin{bmatrix} \frac{\partial \phi_i}{\partial \xi} \\ \frac{\partial \phi_i}{\partial \eta} \\ \frac{\partial \phi_i}{\partial \zeta} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix} \begin{bmatrix} \frac{\partial \phi_i}{\partial x} \\ \frac{\partial \phi_i}{\partial y} \\ \frac{\partial \phi_i}{\partial z} \end{bmatrix} = [J] \begin{bmatrix} \frac{\partial \phi}{\partial x} \\ \frac{\partial \phi_i}{\partial y} \\ \frac{\partial \phi_i}{\partial z} \end{bmatrix}$$

$$\begin{bmatrix} \frac{\partial \Phi_i}{\partial x} \\ \frac{\partial \Phi_i}{\partial y} \\ \frac{\partial \Phi_i}{\partial z} \end{bmatrix} = [J]^{-1} \begin{bmatrix} \frac{\partial \Phi_i}{\partial \xi} \\ \frac{\partial \Phi_i}{\partial \eta} \\ \frac{\partial \Phi_i}{\partial \zeta} \end{bmatrix}$$

$$\underset{\sim}{k} = \int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} \underset{\sim}{\Phi}_e^t \, \underset{\sim}{D}^t \, \underset{\sim}{E} \, \underset{\sim}{D} \, \underset{\sim}{\Phi}_e \det(J) \, d_\xi \, d_\eta \, d_\zeta$$

FIG. 19. Element Stiffness Transformation from the x, y, z Coordinate System to the ξ, u, ζ Coordinate System

NUMERICAL INTEGRATION

The integration indicated in the element stiffness matrix can be quite cumbersome, and most often numerical integration is the only recourse for higher order elements. The basis for deriving most numerical integration quadratures is the approximation of the integrand by a finite degree polynomial which can be integrated readily. For example, consider an integral I defined as

$$I = \int_a^b F(x)dx \tag{114}$$

where the integrand F(x) is complex and cannot be readily evaluated. In such a case the function F(x) can be approximated by an nth degree polynomial as shown

$$F(x) = a_o + a_1 x + \text{-----} + a_n x^n \tag{115}$$

There are (n+1) unknown coefficients (a^s) in the above approximation. A criteria is necessary for evaluation of these coefficients. One such criteria is to require that the approximate function satisfy the exact functional values at (n+1) equal points between the interval a and b. Now the interval a to b is divided equally into n divisions, and the coordinates of the (n+1) points are given by x_o, $x_1, \ldots, x_n$. The criteria can be expressed by a matrix equation

$$\underset{\sim}{F}_e = \underset{\sim}{X}_e \underset{\sim}{A} \tag{116}$$

where $\underset{\sim}{F}_e$ is the vector of function values at the (n+1) points

$$F_e^t = [F_o F_1 F_2 \text{ ----- } F_n] \tag{117}$$

$\underset{\sim}{A}$ is the (n+1) vector of polynomial coefficients

$$A^t = [a_o a_1 a_2 \text{ ----- } a_n] \tag{118}$$

The matrix $\underset{\sim}{X}_e$ is given by

$$\underset{\sim}{X}_e = \begin{bmatrix} 1 & X_o & X_o^2 & . & . & . & X_o^n \\ 1 & X_1 & X_1^2 & . & . & . & X_1^n \\ . & & & & & & . \\ . & & & & & & . \\ . & & & & & & . \\ . & & & & & & . \\ . & & & & & & . \\ 1 & X_n & X_n^2 & . & . & . & X_n^n \end{bmatrix} \tag{119}$$

Now the vector of polynomial coefficients can be determined by

$$\underset{\sim}{A} = \underset{\sim}{X}_e^{-1}\underset{\sim}{F}_e \tag{120}$$

Then the polynomial F(x) can be written as

$$F(x) = \underset{\sim}{X}^t\underset{\sim}{X}_e^{-1}\underset{\sim}{F}_e \tag{121}$$

The polynomial integrand can be integrated with ease. The degree of the polynomial and the associated criteria for determining the coefficients of the polynomial are the limiting factors. There are a number of improvements to this basic concept. Before outlining two of the improved approximations, let us examine an alternative to the above procedure. When the degree of the polynomial approximation is high, finding $\underset{\sim}{X}_e^{-1}$ is not the most convenient or desirable procedure. Instead the alternative is to use nth degree Lagrangian interpolation as in the case of the shape functions. In terms of the Lagrangian interpolation functions, F(x) can be approximated by

$$F(x) = \phi_o F_o + \phi_1 F_1 + \text{-----} + \phi_n F_n \tag{122}$$

where the Lagrangian interpolation functions are given by

$$\phi_i = \frac{\prod_{\substack{j=1 \\ i \neq j}}^{m+1} (x - x_j)}{\prod_{\substack{j=1 \\ j \neq i}}^{m+1} (x_i - x_j)} \quad \textbf{LAGRANGIAN INTERPOLATION FUNCTION} \tag{123}$$

The function values at (n+1) points are $F_o, F_1, \ldots, F_n$. An improvement to the Lagrangian interpolation is the Newton-Cotes interpolation which is written as

$$\int_a^b F(x)dx = (b-a) \sum_{i=o}^{n} C_i^n F_i + R_n \tag{124}$$

where R_n is the remainder. The C_i^n are the Newton-Cotes constants for numerical integration with n sampling points. These constants are given in numerical tables. The interval a to b is divided into n equal segments.

$$h = \frac{b-a}{n} \tag{125}$$

The well known trapezoidal rule of integration corresponds to n=1, and Simpson's rule corresponds to n=2. The accuracy of the Newton's-Cotes approximation can be improved by either increasing the order of approximation or by repeated use of a lower order approximation.

The Gauss-quadrature for numerical integration is a further improvement over both of the above methods. Most finite formulations in practice use Gauss-quadrature and it is given by

$$\int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} F(X,Y,Z)\,dX\,dY\,dZ = \sum_{i=1}^{n}\sum_{j=1}^{n}\sum_{K=1}^{n} H_K F(S_K, S_j, S_i) \tag{126}$$

The coefficients and the integration points are given by

	Abscissa(s)	Weight Coefficient
2	±0.57735026919	1.0
3	±0.77459666924	0.5555555555
	0.0	0.88888888888
4	±0.86113631159	0.34785484514
	±0.33998104358	0.65214515486

The recommended integration order for plane stress isoparametric elements is as follows:

Element	Integration Order Recommended	Maximum Order
4-Node Rectangle Element	2x2	2x2
4-Node Distorted Element	2x2	3x3
8-Node Rectangle Element	2x2	3x3
8-Node Distorted Element	3x3	4x4

A similar recommendation for solid elements is as follows:

Linear Elements	2x2x2
Quadratic and Cubic Elements	3x3x3

ADDITIONAL INFORMATION FOR SOLID ELEMENTS

The stress-strain relations for three dimensional elements with thermal effects is given by

$$\underset{\sim}{\sigma} = \underset{\sim}{G}_e\{\varepsilon - \underset{\sim}{\varepsilon}_t\} \tag{127}$$

where $\underset{\sim}{G}_e$ is the elastic constants matrix and $\underset{\sim}{\varepsilon}_t$ is the thermal strain matrix. The elastic constants matrix for the three-dimensional isotropic case is given by

$$\underset{\sim}{G}_e = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} (1-\nu) & \nu & \nu & 0 & 0 & 0 \\ \nu & (1-\nu) & \nu & 0 & 0 & 0 \\ \nu & \nu & (1-\nu) & 0 & 0 & 0 \\ 0 & 0 & 0 & (1-2\nu)/2 & 0 & 0 \\ 0 & 0 & 0 & 0 & (1-2\nu)/2 & 0 \\ 0 & 0 & 0 & 0 & 0 & (1-2\nu)/2 \end{bmatrix} \tag{128}$$

where E is Young's modulus and ν is Poisson's ratio.

The thermal strain is given by

$$\begin{aligned} \underset{\sim}{\varepsilon}_t = \underset{\sim}{\alpha}_e \underset{\sim}{T} &= [\alpha_x \; \alpha_y \; \alpha_z \; \alpha_{xy} \; \alpha_{xz} \; \alpha_{zx}]^t \underset{\sim}{T} \\ &= [\alpha \; \alpha \; \alpha \; o \; o \; o]\underset{\sim}{T} \end{aligned} \tag{129}$$

The relationship between the element's interior temperature and the nodal temperatures is given by

$$\underset{\sim}{T} = \underset{\sim}{\phi} \underset{\sim}{T}_e \tag{130}$$

$\underset{\sim}{T}_e$ is the vector of temperatures of the grid points to which the element is connected. ϕ is the shape function, and $\underset{\sim}{T}$ is the temperature distribution assumed in the element.

The elastic constants matrix for a three dimensional element made of anisotropic material is given by

$$G_m = \begin{bmatrix} \frac{1-\nu_{23}\nu_{32}}{CE_2E_3} & \frac{\nu_{12}+\nu_{32}\nu_{13}}{CE_1E_3} & \frac{\nu_{13}+\nu_{12}\nu_{23}}{CE_1E_2} & 0 & 0 & 0 \\ & \frac{1-\nu_{31}\nu_{13}}{CE_3E_1} & \frac{\nu_{23}+\nu_{21}\nu_{13}}{CE_1E_2} & 0 & 0 & 0 \\ & & \frac{1-\nu_{12}\nu_{21}}{CE_1E_2} & 0 & 0 & 0 \\ & & & G_{44} & 0 & 0 \\ & \text{SYMMETRIC} & & & G_{55} & 0 \\ & & & & & G_{66} \end{bmatrix} \tag{131}$$

$$C = \frac{1-\nu_{12}\nu_{21}-\nu_{23}\nu_{32}-\nu_{31}\nu_{13}-2\nu_{21}\nu_{32}\nu_{13}}{E_1E_2E_3}$$

where E_1, E_2 and E_3 are the moduli of elasticity in the three material axes direction. The ν's are the Poissons ratios, and G_{44}, G_{55} and G_{66} are the shear moduli in the three material axes directions. The elastic constants can be transformed to any other coordinate system by the transformation indicated in Section 2 and repeated here.

$$\underset{\sim}{G}_e = \underset{\sim}{T}^t \underset{\sim}{G}_m \underset{\sim}{T} \tag{132}$$

where the transformation matrix $\underset{\sim}{T}$ is the same as that given for the strain transformation in Section 2. The thermal coefficient transformation is given by

$$\underset{\sim}{\alpha}_e = \underset{\sim}{T}^{-1} \underset{\sim}{\alpha}_m \tag{133}$$

So far the discussion has centered around the generation of element stiffness matrices, stresses, strains, etc. Once the element information is generated in the local coordinate system, the global stiffness matrix, etc. can be generated by transforming the information from the local coordinate system to the global coordinate system and adding to obtain the master stiffness matrix (see Eq. (54)). To solve for the unknown displacements we still need load matrices (see Eq. (55)). In the case of simple linear elements these load matrices can be generated by a consistent formulation or by a lumping procedure derived from inspection. Either way the results will be within the bounds of the finite element approximation. This is not the case when higher order isoparametric elements are used in the model. Then the recommended procedure is to generate the load matrices consistent with the formulation of the element stiffness matrices. For example, the nodal (grid) forces due to a distributed surface pressure must be obtained by

$$\underset{\sim}{P}_p = \int_s \underset{\sim}{\phi}^t \underset{\sim}{p} ds \tag{134}$$

where $\underset{\sim}{P}_p$ is the vector of grid forces at the discrete grid points of the elements due to the pressure p distributed over the surface. Similarly, the grid forces of a solid element due to distributed body forces can be written as

$$\underset{\sim}{P}_p = \int_v \underset{\sim}{\phi}^t \underset{\sim}{R} dv \tag{135}$$

where $\underset{\sim}{R}$ is the body force vector.

The consistent formulation for the discrete thermal load can be written as

$$\underset{\sim}{P}_T = \int_v \underset{\sim}{C}^t \underset{\sim}{G}_e \underset{\sim}{\alpha}_e \underset{\sim}{\phi} \underset{\sim}{T}_e dv \tag{136}$$

where $\underset{\sim}{C}$ is given by

$$\underset{\sim}{C} = \underset{\sim}{D}\underset{\sim}{\phi} \tag{137}$$

In concluding the solid elements discussion it is worthwhile pointing out the expected output for these elements. In general, most of the solid element information is generated in the basic coordinate system of the element. Then the information can be transformed to the global coordinate system as desired at each grid point.

The element stresses are computed by

$$\underset{\sim}{\sigma} = \underset{\sim}{G}_e \underset{\sim}{D} \underset{\sim}{\phi} \underset{\sim}{u}_e - \underset{\sim}{\alpha}_e \underset{\sim}{\phi} \underset{\sim}{T}_e \tag{138}$$

The stress output for the following elements is generally given at

Linear Element - Eight corner points and at the center

Quadratic and Cubic Elements

* Eight corner points
* Center of each edge
* Center of the element

This output generally consists of

* Principal stresses
* Principal angles
* Mean stress
* Octahedral shear stress

The mean stress or hydrostatic stress is defined as

$$\sigma_n = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z) \tag{139}$$

The octahedral shear stress is defined as

$$\sigma_o = \left[\frac{1}{3}\{(S_x + \sigma_n)^2 + (S_y + \sigma_n)^2 + (S_z + \sigma_n)^2\}\right]^{1/2} \tag{140}$$

where S_x, S_y and S_z are the principal stresses.

6. BENDING ELEMENTS - BARS (BEAMS) AND PLATES

So far the discussion has centered around plane stress (membrane) elements and solid elements. Bending elements are necessary when transverse forces are significant, and when the elements cannot carry these forces by membrane action alone. For example, a wing box constructed out of spars, ribs, skins and rods can transmit the overall loads quite well by membrane action alone, even though the aerodynamic lift forces produce significant bending and shear in the wing box. The bending caused by the aerodynamic lift is resisted by internal couples generated by the tension and compression in the bottom and top skins of the wing where the shear is carried by the spars. However, locally the skins have to carry the pressure load between the substructure supports (spar and ribs) by a bending action. Also compression in the top skin can trigger panel buckling in the top skin. This buckling resistance must also come from the bending action of the plates. So the bending behavior is an important modeling consideration in finite element analysis.

Bars

Among the line elements only the axial force member has been discussed so far. The three dimensional line element which will be referred to as a bar or a beam is the most versatile element in the finite element library. This element can be used very effectively for developing stick models of most aerospace structures. These stick models are very useful in conceptual design for studying the overall dynamic behavior of a vehicle. For example, when details of the overall internal structure are not fully developed (or are not known), the dynamic behavior, such as frequencies, mode shapes, etc., can be adequately predicted for a preliminary assessment of the stability and control characteristics of the vehicle. These elements are easy to model, and they permit rapid parametric studies for improving the handling qualities of the vehicle.

The most general three dimensional bar element has six degrees of freedom at each end. Three of these are translational and three are rotational degrees of freedom. The degrees of freedom of the bar element are shown in Fig. 20.

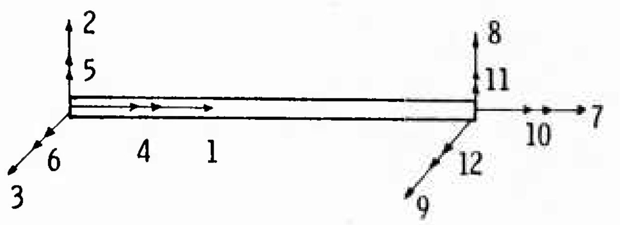


FIG. 20. Degrees of Freedom of the Bar Element

This bar element can resist an axial force (tension or compression in the x-direction), shear in the y and z directions, twisting about its own axis (x-axis) and bending about the y and z axes. These are the six stress-resultants corresponding to the six displacement degrees of freedom at each end. The axial (tension/compression) behavior of the bar can be assumed to be governed by a first order differential equation. Similarly, the torsional behavior is governed by a first order differential equation. The bending and shear behavior in each of the two planes (xy and xz) is governed by one fourth order differential equation. The implicit assumption in the foregoing discussion is that these behaviors are uncoupled. For most beams (cross-sections with at least one axis of symmetry or the reference planes are the principal planes) this assumption is valid. In such a case the element stiffness corresponding to the 12 degrees of freedom can be written by superposition of the four stiffnesses.

$$\underset{\sim}{k} = \underset{\sim}{k}_A + \underset{\sim}{k}_{Bxy} + \underset{\sim}{k}_{Bxz} + \underset{\sim}{k}_T \tag{141}$$

where

$\underset{\sim}{k}$	12x12 Bar element stiffness matrix
$\underset{\sim}{k}_A$	12x12 Axial stiffness contribution
$\underset{\sim}{k}_{Bxy}$	12x12 Bending stiffness in the xy plane
$\underset{\sim}{k}_{Bxz}$	12x12 Bending stiffness in the xz plane
$\underset{\sim}{k}_T$	12x12 Torsional stiffness

all of these matrices are symmetric about the diagonal.

Elements of $\underset{\sim}{k}_A$

$$k_{11} = -k_{14} = k_{44} = \frac{AE}{L} \tag{142}$$

The remaining elements are zero. A is the cross-sectional area of the bar, E is the material modulus of elasticity, and L is the length of the bar.

Elements of $\underset{\sim}{k}_{Bxy}$

$$\begin{aligned} k_{2,2} &= -k_{2,8} = k_{8,8} = r_z = \left[\frac{L}{K_y AG} + \frac{L^3}{12EI_z}\right]^{-1} \\ k_{2,6} &= k_{2,12} = k_{6,8} = -k_{8,12} = \frac{L}{2} r_z \\ k_{6,6} &= k_{12,12} = \frac{L^2}{4} r_z + \frac{EI_z}{L} \\ k_{6,12} &= \frac{L^2}{4} r_z - \frac{EI_z}{L} \end{aligned} \tag{143}$$

The remaining elements are zero. G is the shear modulus of the material. I_z is the moment of inertia of the cross-section about the z-axis, and K_y is the area shear reduction factor in the y-direction.

Elements of $\underset{\sim}{k}_{Bxz}$

$$\begin{aligned} k_{3,3} &= -k_{3,9} = k_{9,9} = r_y = \left[\frac{L}{K_z AG} + \frac{L^3}{12EI_y}\right]^{-1} \\ -k_{3,5} &= -k_{3,11} = k_{5,9} = k_{9,11} = \frac{L}{2} r_y \\ k_{5,5} &= k_{11,11} = \frac{L^2}{4} r_y + \frac{EI_y}{L} \\ k_{5,11} &= \frac{L^2}{4} r_y - \frac{EI_y}{L} \end{aligned} \tag{144}$$

The remaining elements are zero. I_y is the moment of inertia of the cross-section about the y-axis, and k_z is the area shear reduction factor in the z-direction.

Elements of $\underset{\sim}{k}_T$

$$k_{4,4} = -k_{4,7} = k_{7,7} = \frac{GJ}{L} \tag{145}$$

The remaining elements are zero. J is the torsional constant. A list of approximate formulas for the torsional constants is given for popular sections in the Appendix. For most beams the deformations due to shear are small compared to bending, and the terms containing the area shear reduction factors can be neglected. Exceptions are when the bars are very short and their cross-sectional dimensions are large (deep beams). The length referred to in the previous statement is not the element length, but refers to the span of the beam between the supports.

Generally, there are four important cross-sectional properties for bar elements. These are the cross-sectional area (A), the moments of inertia about the y and z axes (I_y, I_z), and the torsional constant (J).

The bar element can be made very versatile by allowing offset provisions at the two ends, by allowing pin flags of up to 5 degrees of freedom at each end, and also by allowing an offset in the elastic and mass C.G. axis. The latter provision allows for the modeling of wing surfaces as beam elements and simulates the bending-torsion coupling behavior. Then the element stiffness needs modification accordingly.

Bending Behavior of Plates

Most plate structures are subjected to both inplane and out of plane forces. The out of plane forces normally induce bending in the plates. If the nonlinear interaction of the inplane and out of plane behavior is small, then these two effects can be uncoupled. The inplane behavior can be represented by membrane elements as discussed earlier. The bending behavior can be derived separately, and the two effects can be superimposed to obtain the combined behavior. This procedure is analogous to the bar element formulation. Except in the case of significant nonlinearities, this procedure is valid for plates made of isotropic materials and symmetric and balanced laminates. For unsymmetric laminates, however, the coupling between the inplane and out of plane behavior cannot be avoided, even when the nonlinear effects are small. This coupling and uncoupling behavior can be explained simply by writing the strain displacement and stress resultant strain curvature relations.

The displacements of a point in the plate can be represented by u, v, w, and they are given by

$$u = u_o - Z\frac{\partial w_o}{\partial x}$$

$$v = v_o - Z\frac{\partial w_o}{\partial y} \tag{146}$$

$$w = w_o$$

where u_o, v_o and w_o are the midplane displacements of the point.

Similarly the strain curvature relations can be written as

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{bmatrix} = \begin{bmatrix} \varepsilon_{x_o} \\ \varepsilon_{y_o} \\ \varepsilon_{xy_o} \end{bmatrix} + Z \begin{bmatrix} k_x \\ k_y \\ k_{xy} \end{bmatrix} \tag{147}$$

The ε_{x_o}, ε_{y_o} and ε_{xy_o} are the mid-surface strains and k_x, k_y and k_{xy} are the plate curvatures. These are are given by

$$\varepsilon_{x_o} = \frac{\partial u_o}{\partial x} \qquad k_x = -\frac{\partial^2 w_o}{\partial x^2}$$

$$\varepsilon_{y_o} = \frac{\partial v_o}{\partial y} \qquad k_y = -\frac{\partial^2 w_o}{\partial y^2} \tag{148}$$

$$\varepsilon_{xy_o} = \frac{\partial u_o}{\partial y} + \frac{\partial v_o}{\partial x} \qquad k_{xy} = -\frac{\partial^2 w_o}{\partial x \partial y}$$

The stress-strain relation in a laminate can be written as

$$\underset{\sim}{\sigma} = \underset{\sim}{Q}\underset{\sim}{\varepsilon} \tag{149}$$

The elastic constants matrix $\underset{\sim}{Q}$ is given by removing the 3rd, 4th and 5th rows and columns from the $\underset{\sim}{G}$ matrix written in Eq. (128).

Now the relation between the stress resultants (inplane forces and moments) and the stretching and curvature strains can be written as

$$\begin{bmatrix} N_x \\ N_Y \\ N_{XY} \\ M_x \\ M_Y \\ M_{XY} \end{bmatrix} = \left[\begin{array}{ccc|ccc} A_{11} & A_{12} & A_{16} & B_{11} & B_{12} & B_{16} \\ A_{12} & A_{22} & A_{26} & B_{12} & B_{22} & B_{26} \\ A_{13} & A_{23} & A_{66} & B_{13} & B_{23} & B_{66} \\ \hline B_{11} & B_{12} & B_{16} & D_{11} & D_{12} & D_{16} \\ B_{12} & B_{22} & B_{26} & D_{12} & D_{22} & D_{26} \\ B_{16} & B_{26} & B_{66} & D_{16} & D_{26} & D_{66} \end{array}\right] \begin{bmatrix} \epsilon_{XO} \\ \epsilon_{YO} \\ \epsilon_{XYO} \\ \hline K_X \\ K_Y \\ K_{XY} \end{bmatrix} \tag{150}$$

where the coefficients A_{ij}, B_{ij} and D_{ij} are given by

$$A_{ij} = \int_{-h/2}^{h/2} Q_{ij}\, dZ \qquad B_{ij} = \int_{-h/2}^{h/2} Q_{ij} Z\, dZ \qquad D_{ij} = \int_{-h/2}^{h/2} Q_{ij} Z^2\, dZ \tag{151}$$

For an isotropic or composite plate with a balanced and symmetric laminate, the $\underset{\sim}{B}$ matrix is zero. As a result the inplane and bending behavior can be uncoupled provided the nonlinear interaction between them is not significant. Based on the above discussion the equivalent stress-strain law for inplane behavior can be written as

$$\begin{bmatrix} \sigma_x \\ \sigma_Y \\ \sigma_{XY} \end{bmatrix} = \frac{1}{h} \begin{bmatrix} A_{11} & A_{12} & A_{13} \\ A_{12} & A_{22} & A_{23} \\ A_{13} & A_{23} & A_{33} \end{bmatrix} \begin{bmatrix} \epsilon_x \\ \epsilon_Y \\ \epsilon_{XY} \end{bmatrix} \tag{152}$$

For bending the equivalent stress-strain law can be written as

$$\begin{bmatrix} \sigma_x \\ \sigma_Y \\ \sigma_{XY} \end{bmatrix} = \frac{12}{h^3} \begin{bmatrix} D_{11} & D_{12} & D_{13} \\ D_{12} & D_{22} & D_{23} \\ D_{13} & D_{23} & D_{33} \end{bmatrix} \tag{153}$$

If the inplane-bending behavior can be uncoupled, then the plate element stiffness can be written as

$$\tilde{k} = \tilde{k}_M + \tilde{k}_B \tag{154}$$

where k_M is the membrane element stiffness matrix derived (for the triangle and the quadrilateral) earlier. However, it was derived only with respect to the inplane displacement degrees of freedom. To expand this matrix to include the bending degrees of freedom, we can simply add rows and columns of zeros. For membrane elements the two degrees of freedom corresponding to the two displacements were specified for each grid point. The degrees of freedom for bending behavior consist of a transverse displacement (normal to the plate) and rotations about the two orthogonal axes in the plane of the plate (see Fig. 21).

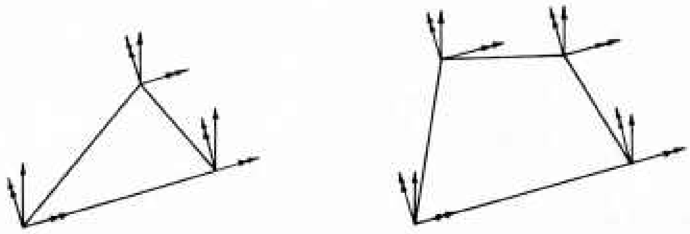


FIG. 21. Degrees of Freedom for the Bending Triangle and Quadrilateral

The stiffness matrix of the bending elements can also be expanded to include the membrane degrees of freedom by adding rows and columns of zeros. When the membrane and bending behaviors are superimposed, the element provides stiffness against five degrees of freedom. The degree of freedom corresponding to rotation about the normal to the plate does not have stiffness, and this fact must be taken into account in order to avoid singularities in the solution of the load deflection equations.

The bending behavior of a plate is governed by a fourth order partial differential equation with the transverse displacement as the dependent variable. The approximating polynomial for the bending element must reflect this fact. Some proposed polynomial approximations for the bending element are listed here.

Triangular Plate Bending Element

$$w(x,y) = a_1 + a_2x + a_3y + a_4x^2 + a_5xy + a_6y^2 + a_7x^3 + a_8(x^2y + xy^2) + a_9y^3 \tag{155}$$

Rectangular Plate Bending Element

$$w(x,y) = a_1 + a_2x + a_3y + a_4x^2 + a_5y^2 + a_6xy + a_7x^3 + a_8y^3 + a_9x^2y + a_{10}xy^2 + a_{11}x^3y + a_{12}xy^3 \tag{156}$$

Similar or variations of these approximations were used by various investigators. Most of these are incomplete polynomial approximations, and there is a certain arbitrariness and controversy. The incompleteness is due to the fact that the number of unknown polynomial coefficients has to be consistent with the number of displacement degrees of freedom assigned to the element. Once the displacement approximations are defined, then the procedure for deriving the element stiffness matrices is similar to that outlined in Sections 4 and 5.

7. MASS PROPERTIES OF THE ELEMENTS FOR DYNAMIC ANALYSIS

In dynamic analysis we need to consider the inertia forces in addition to the elastic forces on the structure. The inertia forces require mass properties. The element mass matrices can be generated by either a lumped mass or a consistent mass approach. The basis for a lumped mass approach is simply by inspection or a linear approximation at best. For example, the distributed mass of a rod can be lumped at each end by half its total mass. Similarly, for a triangular element a third of the mass can be lumped at each grid point and so on. The consistent mass approach on the other hand is derived logically from the kinetic energy of the system and the assumed displacement functions. The procedure is akin to the element stiffness matrices derivations. The term consistent refers to the formulation being consistent with the stiffness derivation.

The kinetic energy of a vibrating (time dependent motion) element can be written

$$T_i = \frac{1}{2} \int_v \rho_i \dot{w}_i^t \dot{w}_i \, dv \tag{157}$$

Substituting the assumed displacement functions in terms of the shape functions and the discrete grid point displacements (see Eq. (70)) into Eq. (157) gives

$$T_i = \frac{1}{2} \dot{v}_i^t m_i \dot{v}_i \tag{158}$$

where the element mass matrix m_i is given by

$$m_i = \int_v \rho_i \phi_i^t \phi_i \, dv \tag{159}$$

The total kinetic energy in the structure is the sum of the kinetic energies of the elements

$$T = \sum_{i=1}^{m} \dot{v}_i^t m_i \dot{v}_i \tag{160}$$

Now a transformation from the local coordinate system to the global system gives

$$T = \dot{u}^t M \dot{u} \tag{161}$$

where u refers to the global displacement vector and M is the global mass matrix and is given by

$$M = \sum_{i=1}^{m} a_i^t m_i a_i \tag{162}$$

The details of the transformation are similar to that indicated for the stiffness matrix in Section 4. Now the equations of dynamic analysis can be derived by substituting the kinetic energy and strain energy terms into Lagrange's equation.

$$M\ddot{u} + C\dot{u} + Ku = P(t) \tag{163}$$

The first term represents the inertia forces (derived from the kinetic energy), the second term is from the dissipative forces, and the third term represents the elastic forces. The term on the right hand side of Eq. (163) represents the applied forces. Equation (163) is a second order matrix dynamic equation of the system, and it (or variations of) is the basis for predicting the dynamic behavior of mechanical systems.

A few important points are worth noting before leaving this section.

* Both the lumped mass approach and the consistent mass approach are acceptable for linear (first order) elements.
* The lumped mass approach has a tendency to overestimate the mass and gives a lower bound approximation to the frequencies.
* The consistent mass approach gives an upper bound approximation to the frequencies as in a Rayleigh-Ritz procedure.
* For higher order isoparametric elements the lumped mass approach is, generally, not acceptable.
* In the case of higher order elements concentrated non-structural masses must be handled judiciously.

8. STRUCTURAL ANALYSIS SOLUTIONS

Finite element analysis problems will be explained here with the help of the analysis equations.

1. Static Analysis

The static structural analysis is represented by the load deflection equations

$$P = Ku \tag{164}$$

where P represents the applied load vector. P may consist of many independent load vectors representing independent flight conditions. The solution of the above equations is generally carried out in three steps.

a. Decomposition: Involves factorization of the stiffness matrix

$$K = LDL^t \tag{165}$$

where $\underset{\sim}{L}$ is a unit lower triangular matrix and $\underset{\sim}{D}$ is a diagonal matrix.

b. Forward Substitution: Involves the solution of $\underset{\sim}{Y}$ from

$$\underset{\sim}{L}\underset{\sim}{Y} = \underset{\sim}{P} \tag{166}$$

c. Back Substitution: Involves the solution of $\underset{\sim}{u}$ from the relation

$$\underset{\sim}{D}\underset{\sim}{L}^t\underset{\sim}{u} = \underset{\sim}{Y} \tag{167}$$

The reason for performing a static structural analysis in three steps is that the procedure takes advantage of the sparseness and symmetric properties of the stiffness matrix.

If there is more than one load vector, only the last two steps have to be repeated. That is, one decomposition and n FBS (Forward and Back Substitutions) are necessary, where n represents the number of load vectors.

Once the load deflection equations are solved (for $\underset{\sim}{u}$), the element displacements can be determined by a simple coordinate transformation (see Section 4). From the element displacements the strains can be determined by the strain-displacement relations. The stresses are then determined by the stress-strain relations.

2. Normal Modes Analysis

The normal modes analysis represents the solution of a free vibration problem. The equations of free vibration analysis are given by

$$\underset{\sim}{M}\ddot{\underset{\sim}{u}} + \underset{\sim}{K}\underset{\sim}{u} = 0 \tag{168}$$

That is no damping or external forces. The solution of this harmonic equation can be written as

$$\underset{\sim}{u} = \underset{\sim}{U}\ \mathrm{Cos}(\omega t + \theta) \tag{169}$$

Substitution of this solution into the dynamic equation gives

$$\omega^2\underset{\sim}{M}\underset{\sim}{U} = \underset{\sim}{K}\underset{\sim}{U} \tag{170}$$

or

$$(\underset{\sim}{K} - \omega^2\underset{\sim}{M})\underset{\sim}{U} = 0 \tag{171}$$

This equation belongs to a class of generalized eigenvalue problems with symmetric matrices. There are many methods for the solution of this problem. These are generally classified into transformation methods and tracking methods. The transformation methods find all the eigenvalues together. The tracking methods, on the other hand, find the desired eigenvalues and eigenvectors only, usually one at a time or in small groups. Some examples of these methods are listed here.

Givens Method	- Transformation Method
Determinant Method Inverse Power Method Subspace Iteration Sturm-Sequence and Bisection	- Tracking Methods

A normal modes analysis is the basic step in most dynamic analyses of large systems (many degrees of freedom), because these systems can only be solved by reducing to a smaller number of equations. This reduction is most effective when the solution is represented by a small set of independent coordinates associated with the normal (natural) modes of the structure.

3. Complex Eigenvalue Analysis

The damped free vibration of a structure can be represented by

$$\underset{\sim}{M}\ddot{\underset{\sim}{u}} + \underset{\sim}{C}\dot{\underset{\sim}{u}} + \underset{\sim}{K}\underset{\sim}{u} = 0 \tag{172}$$

This second order coupled differential equation can be represented in state space in the following form

$$\dot{\underset{\sim}{X}} = \underset{\sim}{A}\underset{\sim}{X} \tag{173}$$

where $\underset{\sim}{X}$ and $\dot{\underset{\sim}{X}}$ are given by

$$\dot{\underset{\sim}{X}} = \left[\begin{array}{c}\ddot{\underset{\sim}{u}}\\ \hline \dot{\underset{\sim}{u}}\end{array}\right] \qquad \underset{\sim}{X} = \left[\begin{array}{c}\dot{\underset{\sim}{u}}\\ \hline \underset{\sim}{u}\end{array}\right] \tag{174}$$

and the plant matrix $\underset{\sim}{A}$ is given by

$$\underset{\sim}{A} = \left[\begin{array}{c|c} -M^{-1}C & -M^{-1}K \\ \hline I & 0 \end{array}\right] \tag{175}$$

The solution of Eq. (173) can be written as

$$\underset{\sim}{A}\underset{\sim}{x} = \lambda \underset{\sim}{I}\underset{\sim}{x} \tag{176}$$

This is a complex eigenvalue problem. The eigenvalues λ will contain real and imaginary parts

$$\lambda = \lambda_R + i\lambda_I \tag{177}$$

The inverse power method and the upper Hessenberg method (similar to Givens) are some of the methods available for the solution of the complex eigenvalue problem.

When the order of the system is very large, a complex eigenvalue analysis can be too expensive. In such cases a modal reduction before a complex eigenvalue analysis is recommended. This modal reduction involves a real eigenvalue analysis (normal modes), and then the full system is expressed in terms of a reduced number of normal coordinates.

REFERENCES

[1] Boresi, A. P., "Elasticity in Engineering Mechanics," Prentice-Hall, Inc., Englewood Cliffs, N.J., 1965.

[2] Zienkiewicz, O. C., "The Finite Element Method in Engineering Science," McGraw Hill Co., London, 1971.

[3] Przemieniecki, J. S., "Theory of Matrix Structural Analysis," McGraw-Hill, New York, 1968.

[4] Cook, Robert D., "Concepts and Applications of Finite element Analysis," John Wiley & Sons, Inc., New York, 1981.

[5] Rao, S. S., "The Finite element Method in Engineering," Pergamon Press, Oxford, England, 1982.

[6] Irons, B. M., "Engineering Application of Numerical Integration in Stiffness Method," AIAA Journal, Vol. 4, 1966, pp. 2035-2037.

[7] Bathe, K. J. and Wilson, E. L., "Numerical Methods in Finite Element Analysis," Prentice-Hall, Inc., Englewood Cliffs, N.J., 1976.

[8] MacNeal, R. H., (Editor), "The NASTRAN Theoretical Manual, Levels 16 and 17," March 1976.

[9] Garvey, S. J., "The Quadrilateral Shear Panel," Aircraft Engineering, May 1951.

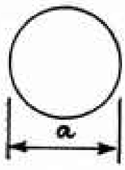
[10] Venkayya, V. B. and Tischler, V. A., "ANALYZE - Analysis of Aerospace Structures with Membrane Elements," AFFDL-TR-78-170.

[11] Venkayya, V. B. and Tischler, V. A., "OPTSTAT - A Computer Program for the Optimal Design of Structures Subjected to Static Loads," Technical Memorandum AFFDL-TM-FBR-79-67.

[12] Sandhu, R. S., "A Survey of Failure Theories of Isotropic and Anisotropic Materials," September 1972, AFFDL-TR-72-71, pp. 19-22.

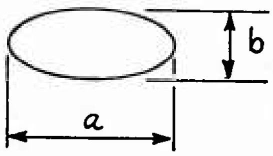
[13] Tsai, S. W., "Strength Characteristics of Composite Materials," April 1965, NASA CR-224, pp. 5-8.

CIRCULAR SECTION



$$J = \frac{\pi a^4}{32}$$

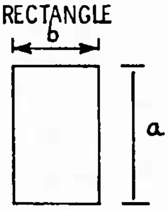
ELLIPTICAL SECTION



$$J = \frac{\pi}{16} \frac{a^3 b^3}{a^2 + b^2}$$

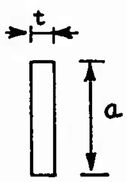
SOLID SECTIONS

RECTANGLE



$$J = \frac{1}{3} b^3 a \left(1 - 0.63 \frac{b}{a}\right)$$

THIN RECTANGLE



$$J = \frac{1}{3} a t^3$$

ARBITRARY SOLID SECTION

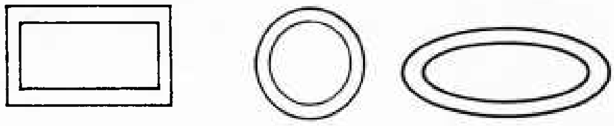
$$J = \frac{\pi}{16} \frac{a^3 b^3}{a^2 + b^2}$$

a AND b ARE EQUIVALENT ELLIPSE DIMENSIONS

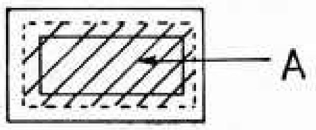
RULES FOR EQUIVALENCE

1. AREA OF ELLIPSE = AREA OF GIVEN SECTION
2. P. I. OF ELLIPSE = P.I. OF GIVEN SECTION

CLOSED SECTIONS

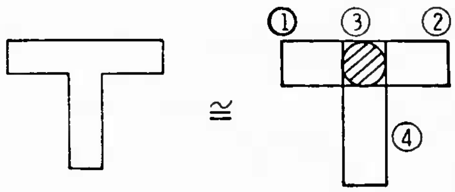


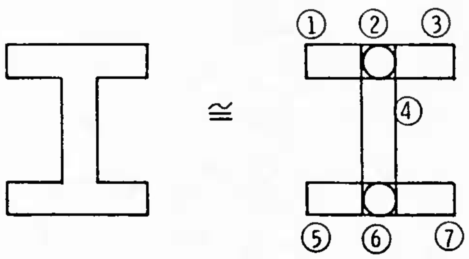
$$J = \frac{4A^2}{\oint \frac{ds}{t}}$$



A = MEAN AREA BOUNDED BY A LINE CONNECTING CENTERS OF THICKNESS

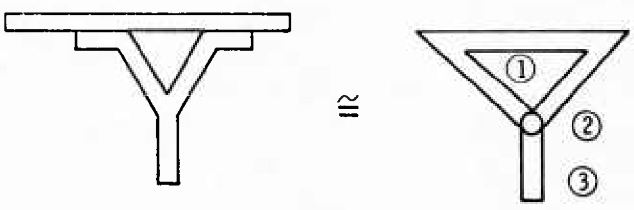
THINWALLED OPEN SECTIONS





+ EFFECT OF WARPING

COMBINED SECTIONS



BIBLIOGRAPHY

This bibliography covering 1978-1986 has been prepared in support of LS-147 using references from the NASA and INSPEC databases by the Royal Signals and Radar Establishment, Malvern, Worcs, UK.

Computation of aeronautical structures (Romanian book)
Calculul structurilor de aviatie
(AA)PETRE, A.
Bucharest, Editura Tehnica, 1984, 368 p. In Romanian. 840000 p. 368 refs 22 In: RO (Romanian) p.316

The book presents both the classical and modern computational methods of aircraft structures. After a brief historical review of aviation, the general design and stress-strain problems are exposed. Aerodynamic principles, loads, shell structures, buckling, reliability, post-buckling behavior and plates and bars are covered in detail. A section on optimization problems ends the classical study. The modern section starts with the matrix methods, discusses the finite element method and concludes with a structural synthesis and optimization theory. The book is of interest to the academic world as well as to engineers and designers working in the aeronautical field.
N.D.

Application of computer-aided structural optimization in the design of aircraft components
Anwendung der rechnergestuetzten Strukturoptimierung bei der Auslegung von Flugzeugbauteilen
(AA)WELLEN, H.
(AA)(Messerschmitt-Boelkow-Blohm GmbH, Bremen, West Germany)
MBB-UT-21-84-OE DGLR, Fachausschussitzung ueber Festigkeit und Bauweisen, Neubiberg, West Germany, May 7, 1984, Paper. 7 p. In German. 840500 p. 7 refs 7 In: GM (German) p.0

In the aerospace industry, the minimization of the structural weight is one of the vital requirements for an economic design of flight vehicles. A computer-aided structural optimization procedure can provide possibilities for performing a weight-optimal dimensioning of structural members in an automatized form, taking into account the employment of programmed, mathematical methods. It is possible to achieve the weight optimum under conditions involving time and cost advantages in comparison to the conventional design process. The Royal Aircraft Establishment (RAE) in England has developed the Structural Analysis and Redesign System (Stars) for a computer-aided structural optimization. Stars makes it possible to solve the involved mathematical problem with the aid of various optimization methods. A description is presented of the modular design of Stars and its operation. The practical application of Stars is discussed, taking into account the solution of design problems related to the Airbus A 310. Attention is given to calculations based on a simplified finite-element model.
G.R.

Advances and trends in structures and dynamics; Proceedings of the Symposium, Washington, DC, October 22-25, 1984
(AA)NOOR, A. K.; (AB)HAYDUK, R. J.
(AA)ED.; (AB)ED.
(AB)(George Washington University, Hampton, VA)
George Washington Univ., Hampton, Va. (GV761922)
Symposium sponsored by George Washington University and NASA. Computers and Structures (ISSN 0045-7949), vol. 20, no. 1-3, 1985, 668 p. For individual items see A85-41102 to A85-41139. 850000 p. 668 In: EN (English) p.0

Among the topics discussed are developments in structural engineering hardware and software, computation for fracture mechanics, trends in numerical analysis and parallel algorithms, mechanics of materials, advances in finite element methods, composite materials and structures, determinations of random motion and dynamic response, optimization theory, automotive tire modeling methods and contact problems, the damping and control of aircraft structures, and advanced structural applications. Specific topics covered include structural design expert systems, the evaluation of finite element system architectures, systolic arrays for finite element analyses, nonlinear finite element computations, hierarchical boundary elements, adaptive substructuring techniques in elastoplastic finite element analyses, automatic tracking of crack propagation, a theory of rate-dependent plasticity, the torsional stability of nonlinear eccentric structures, a computation method for fluid-structure interaction, the seismic analysis of three-dimensional soil-structure interaction, a stress analysis for a composite sandwich panel, toughness criterion identification for unidirectional composite laminates, the modeling of submerged cable dynamics, and damping synthesis for flexible spacecraft structures.
O.C.

The development of efficient models of the deformation of thin-wall structures
O postroenii effektivnykh modelei deformirovaniia tonkostennykh konstruktsii
(AA)OBRAZTSOV, I. F.; (AB)NERUBAILO, B. V.; (AC)ZAITSEV, V. N.; (AD)IVANOV, IU. I.
(AD)(Moskovskii Aviatsionnyi Institut, Moscow, USSR)
Prikladnaia Mekhanika (ISSN 0032-8243), vol. 21, June 1985, p. 61-67. In Russian. 850600 p. 7 refs 9 In: RU (Russian) p.0

Consideration is given to several approaches to developing efficient mathematical thin-wall deformation models and the application of numerical and analytical methods to determine the stress-strain states through discretization of thin-wall structures and through synthesis of stressed states. The study is carried out in the context of the application of the theory of shells and plates to airframe design. The approaches include the finite element method and some analytical methods which are most reliable in determining the stress-strain states.
L.T.

Structures, Structural Dynamics, and Materials Conference, 26th, Orlando, FL, April 15-17, 1985, Technical Papers. Parts 1 & 2
Conference sponsored by AIAA, ASME, ASCE, and AHS. New York, American Institute of Aeronautics and Astronautics, 1985, Pt. 1, 859 p.; pt. 2, 762 p. For individual items see A85-30227 to A85-30405. 850000 p. 1621 In: EN (English) Price of two parts, members, $120.; nonmembers, $150 p.0

Among the topics discussed are sandwich core composite panels, graphite/epoxy composite plates, composite material crack growth behavior, damage tolerance analyses, computer-based structural system design and analysis methods, thermomechanical response prediction, laser irradiation of structures, the buckling behavior of structures, hybrid reinforcing fiber composite characteristics, large space structure antenna design and structural dynamics, multilevel structural optimizations, the fracture behavior of filament-wound structures, and finite element analysis methods. Also covered are metal matrix composite materials, the superplastic forming of high strength aluminum alloys, woven fabric-reinforced composite properties, structural shape optimization, thermal stresses in sandwich panels, airfoil stability and response determination, deployable space structures, space structure control actuators, the stability of flexible structures, structure-borne noise, damping synthesis for large space structures, and optimal vibration control.
O.C.

Introduction to aerospace structural analysis (Book)
(AA)ALLEN, D. H.; (AB)HAISLER, W. E.
(AB)(Texas A&M University, College Station, TX)
New York, John Wiley and Sons, 1985, 518 p. 850000 p. 518 refs 112 In: EN (English) $40 p.0

Aerospace structures are defined as those whose usefulness significantly diminishes with increasing weight; among them may be counted not only aircraft and spacecraft structures, but those of bicycles, ships, and increasingly, those of automobiles. Safety factors are critical in the design of such minimum weight structures. Attention is given, in this comprehensive treatment of the subject for undergraduate students, to fundamental concepts of kinetics, stress, the uniaxial thermomechanical constitution of solids, the multiaxial constitution of elastic and thermoelastic solids, bending and shear in beams, torsion in thin walled closed sections, work and energy principles, the deformation and force analysis of aerospace structures, and finite element stiffness methods.
O.C.

Structural Analysis
Deutsche Forschungs- und Versuchsanstalt fuer Luft- und Raumfahrt, Brunswick (West Germany). (D0696666) Inst. fuer Strukturmechanik.
DFVLR-MITT-84-21; ISSN-0176-7739 841100 p. 365 refs 0 Colloq. held at Brunswick, 4 Jun. 1987 Report will also be announced as translation (ESA-TT-917) In GERMAN and ENGLISH In: AA (Mixed) Avail.: NTIS HC A16/MF A01; DFVLR, Cologne DM 95 p.3120

Coupling of tension and torsion in rods; field consistency in finite element analysis; buckling and post-buckling behavior of shallow shells; optimization of axially compressed carbon fiber reinforced plastic cylinders; a substructure technique applied to fracture mechanics of composites; stress intensity factors as indicators of crack propagation in unidirectional laminates; and static aeroelastic phenomena of composite wings are discussed. For individual titles see N85-29314 through N85-29321.

The application of computer aided structural optimization to the design of aircraft components
ANWENDUNG DER RECHNERGESTUETZTEN STRUKTUROPTIMIERUNG BEI DER AUSLEGUNG VON FLUGZEUGBAUTEILEN
(AA)WELLEN, H.
Messerschmitt-Boelkow-Blohm G.m.b.H., Bremen (West Germany). (MT603998)
MBB-UT-21/84-0 In its Res. and Develop. Tech. and Sci. Repts. 1984 p 61-68 (SEE N85-27724 16-81) 840000 p. 8 refs 0 Presented at DGLR-Fachausschusssitzung Festigkeit u. Bauweisen, Neubiberg, West Germany, 5 Jul. 1984 In: GM (German) Avail: Issuing Activity p. 2873

The Structural Analysis and Redesign System (STARS) program system was used for computer aided structural optimization of aircraft components. Practical use and results, present status, and planned extension of STARS are described. The application of computer aided structural optimization is demonstrated using the inner Airbus A-310 trailing-edge flaps. Computer aided structural optimization offers the possibility of automatic weight-optimal dimensioning of carrying parts using programmed, mathematical methods. The weight optimum leads to time-and cost advantages.
Author (ESA)

Design of load-bearing aircraft structures
Proektirovanie silovykh skhem aviatsionnykh konstruktsii
(AA)KOMAROV, V. A.
IN: Current problems in aviation science and technology (A85-20451 07-01). Moscow, Izdatel'stvo Mashinostroenie, 1984, p. 114-129. In Russian. 840000 p. 16 refs 15 In: RU (Russian) p.918

A method for the computer-aided design of load-bearing structures is proposed which is based on the determination of theoretically optimal structures using continuum models, followed by a graphic analysis of rational pathways of force transmission. Attention is given to the design characteristics of planar structures, wings, fuselages, and arbitrary three-dimensional elastic systems. The development of detailed finite-element methods is considered as the culminating stage in the design of the load-bearing structures.
B.J.

Sonic fatigue design method for the response of CFRP stiffened-skin panels
(AA)HOLEHOUSE, I.
(AA)(Rohr Industries, Inc., Chula Vista, CA)
IN: International Conference on Recent Advances in Structural Dynamics, 2nd, Southampton, England, April 9-13, 1984, Proceedings. Volume 2 (A85-12426 02-39). Southampton, England, University of Southampton, 1984, p. 787-798. 840000 p. 12 refs 7 In: EN (English) p.0

A semi-empirical method for estimating the structural response of carbon fiber-reinforced plastic (CFRP) stiffened-skin panels experiencing random acoustic loading is presented. The technique was developed using experimental and numerical studies. CFRP skin stringer panels were exposed to high intensity noise in a progressive wave tube and finite element analyses characterized the static strains and natural frequencies. Stepwise regression analyses established the empirical relationships between the strain data and panel curvature and aspect ratio. Design equations were selected from the regression equations which most accurately predicted the data. The rms strain and acoustic levels, but not the spectra, were usable for design analyses, yielding strain level accuracies of 9 percent.
M.S.K.

State-of-art and future of aicraft structure design and strength analysis
(AA)YHUANG, Y.; (AB)ZHU, D.
(AB)(Northwestern Polytechnical University, Xian, Shaanxi, People's Republic of China)
Acta Aeronautica et Astronautica Sinica, vol. 5, June 1984, p. 103-111. In Chinese, with abstract in English. 840600 p. 9 refs 10 In: CH (Chinese) p.0

The state of the art and future prospects of design and strength analysis of aircraft structures are briefly reviewed. The subjects discussed include: modern procedures in aircraft structure design, the finite element method and structural analysis programming, mode active control technology and analytical method, the application of fatigue and fracture, advanced composite structures, serious environmental conditions, and reliability analysis for structures.
Author

Predicting structural dynamic behavior using a combined experimental/analytical model (for helicopter design)
(AA)SMITH, M. R.; (AB)WEI, F.-S.
(AA)(Bell Helicopter Textron, Inc., Fort Worth, TX); (AB)(Bell Helicopter Textron, Inc., Fort Worth, TX; Kaman Aerospace Corp., Bloomfield, CT)
N00019-82-G-0009 IN: American Helicopter Society, Annual Forum, 39th, St. Louis, MO, May 9-11, 1983, Proceedings (A84-46326 22-01). Alexandria, VA, American Helicopter Society, 1984, p. 648-655. 840000 p. 8 refs 11 In: EN (English) p.3191

A procedure by means of which to couple an analytically developed helicopter finite element model with experimentally derived structural parameters of wing-mounted store components that can predict overall aircraft system dynamic behavior has been developed. The procedure utilizes an incomplete number of measured mode shapes and modal frequencies to obtain dynamically equivalent stiffness and mass matrices for each externally mounted component. To verify the technique, stiffness and mass matrices are developed for the LAU-68 and Hellfire missile systems, coupled with a finite element model of the U.S. Navy AH-1J helicopter. The results obtained from this method correlate well with actual vibration test data of the total helicopter/store system. This method is valuable for field deployment of a new externally mounted store system which requires integration with a large matrix of existing store configurations. Utilization of this technique should minimize costly flight testing and provide a method to experimentally obtain a mathematical model for highly nonlinear, complex structures.
Author

Computerized methods for analysis and design of aircraft structures
(AA)FREDRIKSSON, B.
(AA)(Saab-Scania, AB, Linkoping, Sweden)
IN: International Council of the Aeronautical Sciences, Congress, 14th, Toulouse, France, September 9-14, 1984, Proceedings. Volume 2 (A84-44926 22-01). New York, American Institute of Aeronautics and Astronautics, 1984, p. 815-827. Research supported by the Forsvaret Materielverk. 840000 p. 13 refs 30 In: EN (English) p.3187

The developments made to utilize computerized methods for analysis and design of aircraft structures are described. The paper discusses the integration of data and methods between geometry, aerodynamics, loads, structures, etc., to form a computer aided engineering environment. The paper is concentrating on structural analysis and sizing. Methods to rationalize the finite element internal loads calculations and to increase the quality of the work is discussed. Methods have been developed to generate local spectra from the finite element result for fatigue and fracture analysis. The rationalization and quality effect of this is discussed. New design criteria and requirements of high performance, light weight and economy implies introduction of new methods for analysis. The paper discusses the developments made in some advanced topics like combined contact and crack problems and structural optimization. The combined use of mini- and supercomputers as well as the twofold effect of rationalization and quality by using supercomputers are discussed.
Author

Structural optimization of an aircraft design with consideration of aeroelastic flutter stability
Strukturelle Optimierung eines Flugzeugentwurfs unter Beruecksichtigung aeroelastischer Flatterstabilitaetsaspekte
(AA)FREYMANN, R.
(AA)(Deutsche Forschungs- und Versuchsanstalt fuer Luft- und Raumfahrt, Institut fuer Aeroelastik, Goettingen, West Germany)
Zeitschrift fuer Flugwissenschaften und Weltraumforschung (ISSN 0342-068X), vol. 8, May-June 1984, p. 208-217. In German. 840600 p. 10 refs 8 In: GM (German) p.2721

A computational algorithm for the minimum-weight optimization of aircraft structures is presented and demonstrated. The method takes both strength-determined scaling criteria and the flutter stability into account and makes use of modal characteristics derived from a reduced finite-element model of the structure. The results of a sample optimization on a cantilevered large-aspect-ratio wing model are presented in tables and graphs, and a flow chart of the algorithm is provided.
T.K.

Possibilities of the SOFEM system from the viewpoint of data generation
Moznosti systemu SOFEM z hlediska generace dat
(AA)MECIAR, M.
Zpravodaj VZLU (ISSN 0044-5355), no. 2, 1984, p. 87-92. In Czech. 840000 p. 6 In: CZ (Czech) p.2936

The paper examines input-data preparation for structural-strength analysis using the FEM-based SOFEM system. Emphasis is placed on data generation with regard to geometric and mechanical characteristics, boundary conditions, and the loading of idealized structures. The possibility of data generation is demonstrated on simple examples.
B.J.

Studies of noise transmission in advanced composite material structures
(AA)ROUSSOS, L. A.; (AB)MCGARY, M. C.; (AC)POWELL, C. A.
National Aeronautics and Space Administration. Langley Research Center, Hampton, Va. (ND210491)
In its ACEE Composite Struct. Technol. p 161-178 (SEE N84-29969 20-24) 830800 p. 19 In: EN (English) Avail.: NTIS HC A09/MF A01 p.3146

Noise characteristics of advanced composite material fuselages were discussed from the standpoints of applicable research programs and noise transmission theory. Experimental verification of the theory was also included.
R.S.F.

NASTRAN analysis of nuclear effects on helicopter transparencies
(AA)LIN, P. T.; (AB)JORGENSON, J. S.
Goodyear Aerospace Corp., Litchfield Park, Ariz. (G2690950)
AD-P003234 In Dayton Univ. Conf. on Aerospace Transparent Mater. and Enclosures p 1083-1095 (SEE N84-26596 17-01) 831200 p. 13 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2597

This paper deals with the linear and geometric nonlinear analysis of the gunner's window on the AH-1S Cobra helicopter in response to a nuclear overpressure environment. The work was sponsored by the Applied Technology Laboratory, U.S. Army Research and Technology Laboratories (AVRADCOM), Fort Eustis, Virginia. Both monolithic stretched acrylic and multilayered transparency configurations are considered in this report. Comparison analyses using both the NASTRAN finite element program and classical Timoshenko plate theory show good agreement. Comparison of the analytical results with experimental observations made by other sources indicates that the geometric nonlinear mathematical models, rather than the linear models, are the more realistic and appropriate representation of transparency response to nuclear overpressure loading in the range considered. It is shown that the classical analysis of a simplified equivalent configuration serves as a useful checkpoint, while finite element programs, such as MSC/NASTRAN, are the necessary analytical tools to examine the complicated configurations and loading conditions.
GRA

Current problems and progress in transparency impact analysis
(AA)BROCKMAN, R. A.
Dayton Univ., Ohio. (DE333333)
AD-P003233 In its Conf. on Aerospace Transparent Mater. and Enclosures p 1057-1082 (SEE N84-26596 17-01) 831200 p. 26 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2597

The design of aircraft transparencies for impact resistance poses a number of difficult problems for the structural analyst. Prominent among these are the accurate modeling of the transparency and its dynamic response, characterization of the construction materials, and evaluation of the applied loadings resulting from soft-body impact. This paper reviews current practices for mathematical modeling of transparency impacts, discusses problem areas in current analysis capabilities, and summarizes some current research on methods for impact simulation.
GRA

Parametric studies of the T-38 student windshield using the finite element of code MAGNA (Materially and Geometrically Nonlinear Analysis)
(AA)NASH, R. A.
Dayton Univ., Ohio. (DE333333)
AD-P003232 F33615-76-C-3103; F33615-80-C-3401 In its Conf. on Aerospace Transparent Mater. and Enclosures p 1040-1055 (SEE N84-26596 17-01) 831200 p. 16 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2597

The parametric studies examine the effect of structural variations on the nonlinear dynamic response of the T-38 student windshield/support structure system to bird impact. The studies were conducted using the MAGNA (Materially and Geometrically Nonlinear Analysis) finite element computer program. Both static and dynamic analyses were performed, examining the effects of changes to the transparency stiffness and intensity of the applied load, both coupled and uncoupled. Significant results of the finite element analysis include transparency deflection peak load versus transparency stiffness, and resultant force plots along the aft arch. A discussion of the application of the finite element method to the birdstrike problem is also presented.
GRA

Simulation of T-38 aircraft student canopy response to cockpit pressure and thermal loads using MAGNA (Materially and Geometrically Nonlinear Analysis)
(AA)MCCARTY, R. E.; (AB)SMITH, R. A.
Air Force Wright Aeronautical Labs., Wright-Patterson AFB, Ohio. (AJ840964)
AD-P003231 In Dayton Univ. Conf. on Aerospace Transparent Mater. and Enclosures p 1009-1039 (SEE N84-26596 17-01) 831200 p. 31 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2597

The linear and nonlinear static response to cockpit pressure and (cold) thermal loads of the forward canopy for the T-38 aircraft has been predicted using the MAGNA (Materially and Geometrically Nonlinear Analysis) finite element computer program. The results obtained are compared to those of earlier analyses and full scale tests. It is concluded that the current canopy design when properly rigged can withstand more than 20 psig pressure, that thermal loads are more critical than cockpit pressure loads, and that providing more attachment fixity at both forward and aft arches would relieve stress concentrations which occur at the canopy corners.
GRA

Analysis of symmetric reinforcement of quasi-isotropic graphite-epoxy plates with a circular cutout under uniaxial tensile loading / M.S. Thesis
(AA)PICKETT, D. H.; (AB)SULLIVAN, P.
Naval Postgraduate School, Monterey, Calif. (NS368219)
AD-A139998 831200 p. 112 In: EN (English) Avail.: NTIS HC A06/MF A01 p.2285

An experimental and computational analysis was made of the strain field around a reinforced circular hole in four HMF330/34 graphite/epoxy (G/Ep) panels under uniaxial tensile loading. The basic panel was a 10.0 in. wide, 26.0 in. long, eight ply, quasi-isotropic (0/+45/90)s cloth laminate. Each panel was reinforced during manufacture by concurring two circular plies of the same material to each side of the panel. A circular one inch hole was drilled concentrically through the laminate to provide a stress concentration. The symmetric reinforcement reported here provided an improvement of 29 to 40% in ultimate strength over a similar but unreinforced panel under the same loading conditions. Test results indicated that panel failure would occur when the fibers in the dominating orientation were strained approximately 1%. There appeared to be a significant load transfer within the laminate at high strains from the failed fibers. A finite element analysis was made and found in excellent agreement with experimental results.
GRA

Residual strength predictions for ballistically damaged aircraft
(AA)CZARNECKI, G. J.
(AA)(USAF, Wright-Patterson AFB, OH)
IN: Annual Mini-Symposium on Aerospace Science and Technology, 9th, Wright-Patterson AFB, OH, March 22, 1983, Proceedings (A83-42526 20-01). New York, American Institute of Aeronautics and Astronautics, 1983, p. 5-2-1 to 5-2-3. 830000 p. 3 In: EN (English) p.3001

In an effort to predict the load carrying capability of ballistically damaged aircraft wings, three MAGNA finite element models have been developed. Wings modeled were the F-4B, and F-15. An A-7 composite wing model is presently under construction. To validate computer predictions, each of the four wings were loaded and ballistically damaged with a high explosive incendiary (HEI) round. Strain gage and deflection data were recorded in the pre- and post-damage conditions. Ultimately, efforts will concentrate toward utilizing this computer code (or a derivation) as a structural design tool which takes survivability/vulnerability aspects into account.
Author

Damped structure design using finite element analysis
(AA)KLUESENER, M. F.; (AB)DRAKE, M. L.
Dayton Univ., Ohio. (DE333333)
In Shock and Vibration Inform. Center The Shock and Vibration Bull., No. 52., Part 5 p 1-12 (SEE N83-30740 19-31) 820500 p. 12 refs 0 In: EN (English) Avail.: NTIS HC A08/MF A01 p.3105

The performance requirements and the life cycle costs for jet engines and aircraft increase the need for functional high cycle fatigue (HCF) control. The methodology of using finite element analysis to evaluate viscoelastic damping treatments for HCF control is presented. Steps for analyzing passive damping treatments are examined. Design criteria used to evaluate the damping applications, as well as two methods of calculating the structural loss factor are discussed. The results from analyses of a stiffened panel and turbine blade are included.
Author

Airworthiness considerations
(AA)HOSKIN, B. C.
Aeronautical Research Labs., Melbourne (Australia). (AF441057)
In its Lectures on Composite Mater. for Aircraft Struct. p 253-262 (SEE N83-30523 19-24) 821000 p. 10 refs 0 In: EN (English) Avail.: NTIS HC A12/MF A01 p.3076

The use of composite materials raises some problems which are different to those for metal aircraft structures. These problems, in turn, raise questions about specific airworthiness requirements for composite aircraft structures. At this state, few such formal specific requirements exist. As an example, consider the matter of the effect of the moisture/temperature environment on structural performance. Although the US Military Standard on Aircraft Structural Integrity states that the standard applies to metallic and non-metallic structures, and although the US Military Handbook details general design procedures for composite structures, neither document specifies a procedure for allowing for environmental effects in the structural integrity program, including the static and fatigue tests on full scale articles. Airworthiness requirements for UK military aircraft containing composite structure only exist in draft form.
Author

Stress intensity factors for two cracks emanating from two holes and approaching each other
(AA)KUO, A. S.; (AB)SAUL, S.; (AC)LEVY, M.
(AC)(Fairchild Republic Co., Farmingdale, NY)
Engineering Fracture Mechanics, vol. 17, no. 3, 1983, p. 281-288. 830000 p. 8 refs 8 In: EN (English) p.496

With reference to the damage tolerance design of aiccraft structures, a numerical method has been developed for examining the interaction between cracks at adjacent holes. A collapsed 1/4 position quadratic quadrilateral isoparametric finite element is used to solve stress intensity factors. The finite element program, called CRACK, calculates the element stiffness matrix with double precision while other calculations are made with single precision. The program can be run in either batch-mode or interactive-mode.
S.C.S.

Optimization of aircraft structures
Optimisation des structures d'avion
(AA)PETIAU, C.; (AB)LECINA, G.
(AB)(Avions Marcel Dassault-Breguet Aviation, Saint-Cloud, Hauts-de-Seine, France)
Journal de Mecanique Theorique et Appliquee, vol. 1, no. 2, 1982, p. 291-309. In French. 820000 p. 19 refs 7 In: FR (French) p.189

A finite element method for weight minimization in the design of aircraft structures is presented. The economic utilization of the method depended on developing an iterative process which was cost-effective. The code that resulted, ELFINI, regrouped, around finite element cores, large branching analyses of the aeronautical structures. Algorithms were devised for linear and nonlinear static constraints, static aeroelasticity, load management, dynamic flutter damping, with transitory and forced response, heat transfer, isothermal mapping, and crack propagation. Partial derivatives are calculated for constraints on the optimization, limits to the flutter speed are defined, and the estremum of transitory response is derived. Two methods for explicit optimization are introduced, and it is noted that final changes, based on small variations in the basic parameters, can be investigated with interactive graphics at the CAD station. An example is presented in terms of designs of a carbon fiber empennage and a delta wing.
M.S.K.

American Helicopter Society, Annual Forum, 38th, Anaheim, CA, May 4-7, 1982, Proceedings
Washington, DC, American Helicopter Society, 1982. 537 p. (For individual items see A82-40506 to A82-40556) 820000 p. 537 In: EN (English) p.3133

Among the topics discussed are the aerodynamics, structural dynamics, propulsion, design, avionics, product assurance, structures and materials, testing, and acoustics of helicopters. The papers presented cover optimum airloads of rotors in hover and forward flight, the evaluation of vertical drag and ground effect, helicopter vibration reduction by rotor blade modal shaping, the finite element analysis of bearingless rotor blade aeroelasticity, adaptive fuel controls, digital full authority engine controls, helicopter autorotation assist concepts, and the conceptual design of an integrated cockpit. Also presented are papers on the demonstration of radar reflector detection, avionics systems for helicopter integration, the adaptation of pultrusion to the manufacture of helicopter components, composite main rotor blades, optimum structural design, the in-plane shear testing of thin panels, error minimization in ground vibration testing, and the prediction of helicopter rotor discrete frequency noise.
O.C.

Composite materials: Mechanics, mechanical properties and fabrication; Proceedings of the Japan-U.S. Conference, Tokyo, Japan, January 12-14, 1981
(AA)KAWATA, K.; (AB)AKASAKA, T.
(AA)(ED.)
(AA)(Tokyo, University, Tokyo, Japan); (AB)(Chuo University, Hachioji, Tokyo, Japan)
Conference sponsored by the Japan Society for Composite Materials and Nihon Itagarasu Zairyokogaku Joseikai. Barking, Essex, England, Applied Science Publishers, 1982. 575 p (For individual items see A82-39852 to A82-39897) 820000 p. 575 In: EN (English) $68 p. 3054

This conference on composite materials opens with consideration of such topics in dynamic behavior and wave propagation as the impact resistance and dynamic analysis of composites, wave propagation in a composite cylinder, and transient wave propagation in a viscoelastic laminate. It then proceeds to stress analysis and mechanical properties, including the equivalent inclusion method, elastic constants and internal friction in composites, finite element method and photoelasticity analyses, fiber orientation, and damping properties. Also covered are composite fatigue and fracture properties, viscoelasticity, elastoplastic fracture toughness, metal matrix composites, ceramic and rubber composites, thermal and environmental problems, the strength of composite structural elements, composite structure design methods and prospective composite applications in aircraft structures, and educational methods for composite materials engineering.
O.C.

Static and aeroelastic optimization of aircraft
Optimisation statique et aeroelastique des avions
(AA)THOMAS, J. M.
(AA)(Societe Nationale Industrielle Aerospatiale, Toulouse, France)
Deutsche Forschungs- und Versuchsanstalt fuer Luft- und Raumfahrt, International Symposium on Aeroelasticity, Nuremberg, West Germany, Oct. 5-7, 1981, Paper. 9 p. In French. 811000 p. 9 In: FR (French) p.2847

Techniques for the finite element modeling of the static and aeroelastic stresses on an aircraft are described. The methods are intended to serve to define sustainable loads in aircraft structures which are built with minimized weight. Attention is focussed on the components of the ASELF program, which comprises analyses of linear and nonlinear statics, crack propagation, thermal effects and static and dynamic aeroelastic characteristics. Partial derivatives are obtained for the displacements, elastic, principal, and equivalent stresses, the rupture criteria, panel buckling, and internal interactions between groups of elements. Constraints are placed on the optimization through formulation of the displacements, a rigidity matrix, and a determination of the external stresses. An optimization example is provided for a composite sheets bordering openings in the aircraft or in the empennage using 1952 degrees of freedom with 3684 elements.
M.S.K.

Structures, Structural Dynamics and Materials Conference, 23rd, New Orleans, LA, May 10-12, 1982, Collection of Technical Papers. Part 1 - Structures and materials. Part 2 - Structural dynamics and design engineering
Conference sponsored by AIAA, ASME, ASCE, and AHS. New York, American Institute of Aeronautics and Astronautics, 1982. Pt. 1, 532 p.; pt. 2, 645 p. (For individual items see A82-30077 to A82-30192) 820000 p. 1177 In: EN (English) PRICE OF TWO PARTS, MEMBERS, $100.; NONMEMBERS, $125 p.2105

An integration scheme to determine the dynamic response of a launch vehicle with several payloads is considered along with aeroelastic characteristics of the Space Shuttle external tank cable trays, the structural design of integral tankage for advanced space transportation systems, and optimum damping locations for structural vibration control. Attention is given to a damage induced aeroelastic failure mode involving combination and parametric resonant instabilities of lifting surfaces, passive damping mechanisms in large space structures, an automated technique for improving modal test/analysis correlation, pressure measurements on twin vertical tails in buffeting flow, and a wind-tunnel study of the aerodynamic characteristics of a slotted versus smooth-skin supercritical wing. Other topics explored are related to the active control of aeroelastic divergence, stress constraints in optimality criteria design, and damage tolerant design using collapse techniques.
G.R.

Structural system identification technology verification / Final Report
(AA)GIANSANTE, N.; (AB)BERMAN, A.; (AC)FLANNELLY, W. G.; (AD)NAGY, E. J.
Kaman Aerospace Corp., Bloomfield, Conn. (KC616275) A2024546
AD-A109181; R-1631; USAAVRADCOM-TR-81-D-28 DAAK51-78-C-0017; DA PROJ. 1L1-62209-AH-76 Army Research and Technology Labs. Ft. Eustis, Va. 811100 p. 218 refs 0 In: EN (English) Avail.: NTIS HC A10/MF A01 p.1376

Structural system identification is the method of obtaining structural and dynamic mathematical models and improving existing mathematical models using ground vibration test data. The purpose of the subject program was to perform experimental, development, and research work to verify the concepts of structural system identification technology. To accomplish this, system identification techniques were applied to a U. S. Army AH-1G helicopter fuselage to create a mathematical model from ground vibration test data, to improve a reduced model of an existing NASTRAN model of the AH-1G using shake test data, and to test the effectiveness of these new mathematical models in predicting the effects of stiffness and mass changes to the airframe. The results of the program indicate that system identification is a viable and cost-effective technique for developing new models and for improving existing finite-element models of an airframe using ground vibration test data.
Author (GRA)

Stiffness degradation of impact damaged structure
Advisory Group for Aerospace Research and Development, Neuilly-Sur-Seine (France). (AD455458)
In its Design Manual for Impact Damage Tolerant Aircraft Struct. p 159-160 (SEE N82-17160 08-05) 811000 p. 2 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1029

Stiffness reduction in impact damaged structural elements can be important from two standpoints. The first is the alteration of load distribution within the structure, potentially causing overloading and failure of undamaged elements. The second area of potential concern is the residual stiffness of major structural components, since stiffness degradation may lead to instability and control inadequacy. These two topics are discussed, however, there are few verified analysis methods for predicting stiffness degradation associated with ballistic damage. Both of these stiffness degradation effects, but particularly the latter, become increasingly significant as the extent of the inflicted damage becomes larger. Stiffness degradation may well be a problem for HE projectile impacts, but it is generally insignificant with small arms.
T.M.

Lighter-Than-Air Systems Technology Conference, Annapolis, MD, July 8-10, 1981, Collection of Technical Papers

Conference sponsored by the American Institute of Aeronautics and Astronautics. New York, American Institute of Aeronautics and Astronautics, Inc., 1981. 160 p. (For individual items see A81-38527 to A81-38545) 810000 p. 160 In: EN (English) MEMBERS, $25; NONMEMBERS, $35 p.2871

Among the topics discussed are: regional development of LTA systems, such as in Latin America and Great Britain; the design and development of a thermal airship, and a status report on the Solar Powered Stratospheric Platform (SPSP); and aerodynamic studies on laminar flow, heavy lift airship dynamics, estimation techniques, survivability in atmospheric turbulence, and predictive steering control. Also covered are such structural design topics as rigid-pressure airships, tethered aerostat dynamics, bulkheads, and finite element analysis of flexible aerostat structures.
O.C.

Sonic fatigue design techniques for graphite-epoxy stiffened-skin panels

(AA)HOLEHOUSE, I.

(AA)(Rohr Industries, Inc., Chula Vista, Calif.)

AIAA 81-0633 In: Structures, Structural Dynamics and Materials Conference, 22nd, Atlanta, Ga., April 6-8, 1981, and AIAA Dynamics Specialists Conference, Atlanta, Ga., April 9, 10, 1981, Technical Papers. Part 2. (A81-29428 12-01) New York, American Institute of Aeronautics and Astronautics, Inc., 1981, p. 574-579. USAF-sponsored research. 810000 p. 6 refs 9 In: EN (English) p.2023

A combined analytical and experimental program was conducted in order to develop a semi-empirical sonic fatigue design method for curved and flat graphite-epoxy skin-stringer panels. A range of multi-bay panels was subjected to high intensity noise environments in a progressive-wave tube. Shaker tests were also performed in order to provide additional random fatigue data. Finite-element analyses were carried out on the test panel designs, generating static strains and frequencies. Multiple stepwise regression analysis was used to develop the sonic fatigue design method. Design equations and a nomograph are presented. Comparisons of sonic fatigue resistance between graphite and aluminum panels were also carried out. The design method developed is presented as a self-contained section and is suitable for practical design use.
(Author)

Structural flight loads simulation capability. Volume 2: Structural analysis computer program user's manual / Final Report, Aug. 1977 - Sep. 1980

(AA)BRUNER, T. S.; (AB)BOUCHARD, M. P.; (AC)GEBARA, J. G.; (AD)HECHT, M. J.; (AE)BOGNER, F. K.

Dayton Univ., Ohio. (DE333333) Aerospace Mechanics Div. AJ840964

AD-A096594; UDR-TR-80-73-VOL-2; AFWAL-TR-80-3118-VOL-2 F33615-76-C--3135 AFWAL Wright-Patterson AFB, Ohio 801100 p. 358 refs 0 In: EN (English) Avail.: NTIS HC A16/MF A01 p.1718

A complete system for the modeling, analysis and post-analysis of wing structures utilizing finite elements in simulated flight loads testing has been developed. The preprocessor incorporated the MAGNA element types 3, 4 and 5 (2-D membrane, truss and thin shells) into three predefined wing class models and allows for the conversion of existing wing models to be analyzed by MAGNA. MAGNA is a very powerful and flexible material and geometrical nonlinear analysis program capable of solving a wide variety of finite element problems. Two postprocessors are coupled to the modeling and analysis of the wing structures to provide model geometry, stress or strain contour or relief displacement plots of the model and analysis results.
GRA

Engineering application of the finite element method; Proceedings of the International Conference, Hovik, Norway, May 9-11, 1979

(AA)BOMAN, P.

(AA)(ED.)

(AA)(Computas A/S, Hovik, Norway)

Conference sponsored by the Norske Veritas. Computers and Structures, vol. 12, Oct. 1980. 311 p. (For individual items see A81-16298 to A81-16309) 801000 p. 311 In: EN (English) p.558

The conference covered topics including finite element study of hull-deckhouse interaction, modeling of finite element analysis of ship vibration, a spectral method for estimating the reliability of reactor containment, and linear and nonlinear three-dimensional thermal analysis of a radial gas turbine wheel. Also discussed are fluid-structure interaction, calculation of critical flutter speeds of an aircraft in subsonic flow, algorithms for nonlinear structural problems, stability analysis of cylindrical shells, and applications of adaptive mesh refinement.
A.T.

Structural optimization with static and aeroelastic constraints
(AA)MATHIAS, D.; (AB)ROEHRLE, H.; (AC)ARTMANN, J.
Dornier-Werke G.m.b.H., Friedrichshafen (West Germany). (D0425275) Structural Dynamics Dept.
In AGARD The Use of Computers as a Design Tool 11 p (SEE N80-21243 12-01) 800100 p. 11 refs 0 In: EN (English) Avail.: NTIS HC A19/MF A01 p.1512

An optimization program is presented which is based on the finite element method and, within the actual optimization step, works according to the gradient method. The DYNOPT computer program was applied to a clamped straight wing. The wing was statically loaded and had eccentric masses and rotational inertias representing rudders and actuators. These eccentricities ensured the coupling between the bending and torsional deformations. The minimum weight of the structure was obtained after 15 iteraction steps while all boundary conditions were observed.
R.C.T.

Application of finite element analysis to derivation of structural weight
(AA)HUTTON, J. G.; (AB)RICHMOND, L. D.
(AB)(Boeing Aerospace Co., Seattle, Wash.)
SAWE PAPER 1271 Society of Allied Weight Engineers, Annual Conference, 38th, New York, N.Y., May 7-9, 1979, 29 p. 790500 p. 29 refs 7 In: EN (English) p.1027

The paper presents application of finite element analysis to derive methodology for estimation of structural system weight. The study developed factoring logic and its testing, and the documentation of interdisciplinary interaction for model development. The numerical factors were composed of subfactors that accounted for modeling technique, construction method, material, and installation details. The F-15A was used as the known structural system for testing of the weight factor logic; a finite element model was developed for the wing box, and a simplified beam body and horizontal tail were included for simulation of the wing support and to provide balanced aircraft loads. The correlation of the factored finite element and as-built weights was good for the cover panels; the total cover weight compared within 3% with a plus or minus 10% spread among the individual panels.
A.T.

Optimal design of wing structures with substructuring
(AA)GOVIL, A. K.; (AB)ARORA, J. S.; (AC)HAUG, E. J.
(AA)(Iowa, University, Iowa City, Iowa; M.N.R. Engineering College, Allahabad, India); (AC)(Iowa, University, Iowa City, Iowa)
Computers and Structures, vol. 10, Dec. 1979, p. 899-910. Army-supported research. 791200 p. 12 refs 17 In: EN (English) p.4544

An iterative method for optimal design of large scale structures that incorporates the concept of substructuring is extensively applied to wing-type structures to demonstrate its generality, effectiveness and efficiency. Optimum designs for several wing-type structures are obtained and compared with results available in the literature. It is shown that considerable efficiencies can be achieved by integration of the substructuring concept into a structural optimization algorithm.
(Author)

Wing center section optimization with stress and local instability constraints
(AA)ISREB, M.
(AA)(Riyadh, University, Riyadh, Saudi Arabia)
Computers and Structures, vol. 10, Dec. 1979, p. 855-861. 791200 p. 7 refs 13 In: EN (English) p.4543

This paper considers the optimization of bar/shear panel/unidirectionally stiffened panel idealization finite element model of the wing center section of an airplane. The finite element model is synthesized with respect to stress and local instability constraints. The paper introduces more realistic and flexible synthesis approach over the previous published work.
(Author)

Application of numerical methods to heat transfer and thermal stress analysis of aerospace vehicles
(AA)WIETING, A. R.
(AA)(NASA, Langley Research Center, Hampton, Va.)
National Aeronautics and Space Administration. Langley Research Center, Hampton, Va. (ND210491)
International Conference on Numerical Methods in Thermal Problems, Swansea, Wales, July 2-6, 1979, Paper. 11 p. 790700 p. 11 refs 9 In: EN (English) p.4351

The paper describes a thermal-structural design analysis study of a fuel-injection strut for a hydrogen-cooled scramjet engine for a supersonic transport, utilizing finite-element methodology. Applications of finite-element and finite-difference codes to the thermal-structural design-analysis of space transports and structures are discussed. The interaction between the thermal and structural analyses has led to development of finite-element thermal methodology to improve the integration between these two disciplines. The integrated thermal-structural analysis capability developed within the framework of a computer code is outlined.
V.T.

Design of advanced titanium structures (for Advanced Tactical Systems aircraft fuselage)
(AA)PAEZ, C. A.; (AB)GORDON, R.
(AB)(Grumman Aerospace Corp., Bethpage, N.Y.)
AIAA PAPER 79-1805 F33615-77-3109 American Institute of Aeronautics and Astronautics, Aircraft Systems and Technology Meeting, New York, N.Y., Aug. 20-22, 1979, 12 p. 790800 p. 12 In: EN (English) p.3881

An application of superplastic forming/diffusion bonding (SPF/DB) techniques to a Mach 2.0-class aircraft (ATS/BLAST) is discussed. Consideration is given to high-temperature environment due to aerodynamic and engine heating; loads and dynamic analyses; fracture mechanics analyses of SPF/DB structures; detail design of major components; and concept evaluation. Emphasis is placed on maintainability and reliability assessment and weight and cost analyses of advanced concepts. Optimization techniques, joint details, and design/process/tooling interactions are outlined. Attention is given to potential problem areas and future applications of this new design/fabrication method.
V.T.

Design against fatigue - Current trends (for aircraft structural reliability)
(AA)KIRKBY, W. T.; (AB)FORSYTH, P. J. E.; (AC)MAXWELL, R. D. J.
(AC)(Royal Aircraft Establishment, Farnborough, Hants., England)
AIAA 79-0689 In: Atlantic Aeronautical Conference, Williamsburg, Va., March 26-28, 1979, Technical Papers. (A79-27351 10-05) New York, American Institute of Aeronautics and Astronautics, Inc., 1979, p. 27-39. 790000 p. 13 refs 6 In: EN (English) p.1745

The current state-of-the-art in design against fatigue of aircraft structures is reviewed. An outline is given of the design philosophies which have evolved to meet the fatigue problem over the past three decades. Problems of design and operation are summarized with reference to fatigue loads, design trends and developments (e.g. active control technology and in-service load measurement), design methods (e.g., the finite element method), and fatigue design with metallic as well as with composite materials.
B.J.

Substructuring methods for design sensitivity analysis and structural optimization / Interim Technical Report, May - Aug. 1977
(AA)GOVIL, A. K.; (AB)ARORA, J. S.; (AC)HAUG, E. J.
Iowa Univ., Iowa City. (I4630032) Materials Engineering Div.
AD-A065935; TR-34 DAAK11-77-C-0023; DAAA09-76-C-2013 770800 p. 229 refs 0 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1931

This report presents an iterative method for optimal design of large scale structures that incorporates the concept of substructuring. Design sensitivity analysis for the method is developed in a state space setting, in which the symmetry of the structural stiffness matrix is utilized to define efficient adjoint calculations that yield explicit design derivatives. The entire procedure is then presented as a convenient computational algorithm. Applications of the method are given for optimal design of two and three dimensional truss, idealized wing, and framed structures. Computer programs based on the present algorithm are presented for three truss structures (10 member plane cantilever truss, 200 member plane truss, 63 member space truss), three idealized wing structures (18 element wing box beam, 39 element rectangular wing, 150 element swept wing), and three framed structures (one-bay two-story plane frame, two-bay six-story plane frame, 48 element space frame). Results obtained with the substructing formulation are compared first with results obtained without substructing and then with results obtained with other methods.
GRA

Fracture Mechanics Design Methodology (aircraft structures)
Advisory Group for Aerospace Research and Development, Paris (France). (AD481245)
AGARD-LS-97; ISBN-92-835-1294-4; AD-A066808 790100 p. 236 refs 0 AGARD lecture series presented at Delft, The Netherlands, 5-6 Oct. 1978; Munchen, Germany, 9-10 Oct. 1978; Sacavem, Portugal, 12-13 Oct. 1978 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1435

The state of the art of the application of fracture mechanics to the fail safety and damage tolerance assessment of aircraft structures is examined. Basic principles of fracture mechanics are reviewed. It is shown that although damage assessment analysis has passed the stage where tests were the only means to get answers to pertinent questions regarding crack growth and residual strength, tests are still indispensable. For individual titles, see N79-20140 throuah N79-20420.

Application of an interactive graphics system for the design and optimization of aircraft lifting surfaces
(AA)DROR, B.; (AB)EMIL, S.; (AC)BURNS, J.; (AD)KALMAN, H.
(AD)(Israel Aircraft Industries, Ltd., Lod, Israel)
In: Symposium on Applications of Computer Methods in Engineering, Los Angeles, Calif., August 23-26, 1977, Proceedings. Volume 2. (A79-12401 02-31) Los Angeles, University of Southern California, 1978, p. 787-797. 780000 p. 11 refs 9 In: EN (English) p.185

Application of a computerized procedure for the design and optimization of aircraft lifting surface structures employing interactive graphics is described. The procedure represents a subsystem of the multidisciplinary Interactive Structural Sizing and Analysis System (ISSAS), currently being used at IAI. The paper describes the modules that deal with (1) the design of the primary structural layout, (2) the finite element modeling, (3) the analysis and optimization and (4) the reduction of analysis results. The procedure described herein represents a sizable saving in elapsed time and manhours per design iteration relative to conventional methods, and provides for the design of a superior quality structure in a shorter period of time and at lower cost.
(Author)

Decreasing stress concentrations in structures made of high-strength materials
K voprosu o snizhenii kontsentratsii napriazhenii v konstruktsiiakh iz vysokoprochnykh materialov
(AA)GALKINA, N. S.; (AB)GRISHIN, V. I.
TsAGI, Uchenye Zapiski, vol. 8, no. 1, 1977, p. 148-151. In Russian. 770000 p. 4 refs 6 In: RU (Russian) p.249

A method of designing for minimal local stresses in a main frame prepared from high-strength aluminum (or steel) alloys is proposed. Stresses are determined by the finite element method, within the framework of small elastoplastic deformation theory. The relations used for writing a program for computing the stress-strain state of a structure in the case of elastic and inelastic behavior of the material are examined.
V.P.

Conference on Helicopter Structures Technology, Moffett Field, Calif., November 16-18, 1977, Proceedings
Conference sponsored by the American Helicopter Society and NASA. Moffett Field, Calif., U.S. Army Air Mobility Research and Development Laboratory, 1978. 211 p (For individual items see A79-10904 to A79-10921) 780000 p. 211 In: EN (English) $10.00 p.13

Work on advanced concepts for helicopter designs is reported. Emphasis is on use of advanced composites, damage-tolerant design, and load calculations. Topics covered include structural design flight maneuver loads using PDP-10 flight dynamics model, use of 3-D finite element analysis in design of helicopter mechanical components, damage-tolerant design of the YUH-61A main rotor system, survivability of helicopters to rotor blade ballistic damage, development of a multitubular spar composite main rotor blade, and a bearingless main rotor structural design approach using advanced composites.
P.T.H.

Minimum weight design of stiffened panels with fracture constraints
(AA)DOBBS, M. W.; (AB)NELSON, R. B.
(AB)(California, University, Los Angeles, Calif.)
AF-AFOSR-74-2460A Computers and Structures, vol. 8, June 1978, p. 753-759. 780600 p. 7 refs 22 In: EN (English) p.4029

An efficient optimality criteria method is presented for the automated minimum-weight design of structural components for which analytical solutions for developed stress intensity factors are not available. The inclusion of fracture constraints in the automated design process is a logical extension of present structural optimization methods which include stress, displacement, buckling, frequency and aeroelastic flutter constraints. The finite element method is used for stress analysis, while the strain energy release method (the compliance method) is employed to calculate developed opening mode stress intensity factors. Only two structural analyses are needed at each design iteration to calculate the necessary response gradient information and the developed stress intensity factor. The structure is iteratively resized to satisfy the Kuhn-Tucker necessary conditions for a local optimum design. The design of a flat stiffened panel approximating an aircraft fuselage panel is presented.
S.D.

Analysis of semimonocoque beam sections by the displacement method
(AA)MANTEGAZZA, P.
(AA)(Milano, Politecnico, Milan, Italy)
L'Aerotecnica - Missili e Spazio, vol. 56, Dec. 1977, p. 179-182. 771200 p. 4 refs 9 In: EN (English) p.2547

Displacements and rotations of a semimonocoque beam structure of moderate taper and sweep are studied through use of a simply implemented program for digital computers. Adoption of warping displacements and torsion deformations as the primary unknowns permits development of an efficient analytical program for the aircraft fuselage structure. The displacement formulation employed in the analysis is a special version of the general warping function method used to solve torsion and shear problems for arbitrary sections.
J.M.B.

Interactive design and optimization of flight-vehicle lifting surface structures
(AA)DROR, B.; (AB)EMIL, S.; (AC)BURNS, J.; (AD)KALMAN, H.
(AD)(Israel Aircraft Industries, Ltd., Lod, Israel)
Computers and Structures, vol. 8, June 1978, p. 657-665. 780600 p. 9 refs 15 In: EN (English) p.4028

A computerized procedure, including low-cost time-shared graphic display terminals, for the interactive sizing and optimization of flight-vehicle lifting-surface structures is described. The modules pertaining to the interactive structural design, finite-element modeling and optimization features are discussed. They are: (1) a module for graphic layout, design and modification of lifting-surface structures; (2) one for automated-interactive graphics generation, visual checkout and modification of three-dimensional structural analysis finite element models; (3) one for analysis and optimization; and (4) another for interactive graphics visualization of structural analysis results. The use of low-cost graphics terminals in conjunction with centralized data bases is the key to ensuring sizable savings in design time and cost with a high-quality structure.
S.D.

The evaluation of several finite elements for the calculation of wing structures
Valutazione di alcuni elementi finiti per il calcolo di strutture alari
(AA)BORRI, M.; (AB)CARDANI, C.
(AB)(Milano, Politechnico, Milan, Italy)
In: Associazione Italiana di Aeronautica e Astronautica, National Congress, 3rd, Turin, Italy, September 30-October 3, 1975, Proceedings. Volume 1. (A78-19026 06-01) Turin, Libreria Editrice Universitaria Levrotto e Bella, 1975, p. 291-300. In Italian. 750000 p. 10 refs 6 In: IT (Italian) p.990

Plane-stress isoparametric triangular elements with six nodes, used in conjunction with axial elements having three nodes, are compared with several other types of elements (three-node triangular, rectangular, and longitudinal) adopted for finite element calculations of wing structures. The comparison involves analysis of a typical rectangular box structure with a missing section subjected to various loading conditions. The isoparametric triangular elements, in conjunction with the axial elements, is found to offer an efficient and widely applicable analytical technique.
J.M.B.

The AV-8B fatigue test program
(AA)NEWMAN, N. D.
(AA)(McDonnell Aircraft Co., St. Louis, MO)
IN: Helicopter Fatigue Specialists' Meeting, St. Louis, MO, October 16-18, 1984, Proceedings (A86-18430 06-01). Alexandria, VA, American Helicopter Society, 1984, p. 14-0 to 14-27. 840000 p. 28 In: EN (English) p.0

A structural fatigue test program was conducted over 1982-1984 as part of the AV-8B Full Scale Development Program which encompassed a full scale airframe fatigue test and auxiliary fatigue tests of both metallic and composite structural components. The 12,000 hours of severe airframe use thus simulated were equivalent to two service lifetimes. Attention is given to the AV-8B's fatigue design philosophy, load spectra, fatigue test methods, and test results, with attention to problem resolution.
O.C.

Three-dimensional elastic-plastic finite-element analysis of fatigue crack propagation / Final Report, 1 Jun. - 1 Nov. 1985
(AA)GOGLIA, G. L.; (AB)CHERMAHINI, R. G.
Old Dominion Univ., Norfolk, Va. (OS853217) Dept. of Mechanical Engineering and Mechanics.
NASA-CR-176415; NAS 1.26:176415 NAG1-529 851100 p. 60 refs 0 In: EN (English) Avail.: NTIS HC A04/MF A01 p.787

Fatigue cracks are a major problem in designing structures subjected to cyclic loading. Cracks frequently occur in structures such as aircraft and spacecraft. The inspection intervals of many aircraft structures are based on crack-propagation lives. Therefore, improved prediction of propagation lives under flight-load conditions (variable-amplitude loading) are needed to provide more realistic design criteria for these structures. The main thrust was to develop a three-dimensional, nonlinear, elastic-plastic, finite element program capable of extending a crack and changing boundary conditions for the model under consideration. The finite-element model is composed of 8-noded (linear-strain) isoparametric elements. In the analysis, the material is assumed to be elastic-perfectly plastic. The cycle stress-strain curve for the material is shown Zienkiewicz's initial-stress method, von Mises's yield criterion, and Drucker's normality condition under small-strain assumptions are used to account for plasticity. The three-dimensional analysis is capable of extending the crack and changing boundary conditions under cyclic loading.
Author

Optimization of airplane wing structures under gust loads
(AA)RAO, S. S.
(AA)(San Diego State University, CA)
Computers and Structures (ISSN 0045-7949), vol. 21, no. 4, 1985, p. 741-749. 850000 p. 9 refs 12 In: EN (English) p.0

A methodology is presented for the optimum design of aircraft wing structures subjected to gust loads. The equations of motion, in the form of coupled integro-differential equations, are solved numerically and the stresses in the aircraft wing structure are found for a discrete gust encounter. The gust is assumed to be one minus cosine type and uniform along the span of the wing. In order to find the behavior of the wing structure under gust loads and also to obtain a physical insight into the nature of the optimum solution, the design of the typical section (symmetric double wedge airfoil) is studied by using a graphical procedure. Then a more realistic wing optimization problem is formulated as a constrained nonlinear programming problem based on finite element modeling and the optimum solution is found by using the interior penalty function method. A sensitivity analysis is conducted to find the effects of changes in design variables about the optimum point on the response quantities of the wing structure.
Author

Israel Annual Conference on Aviation and Astronautics, 26th, Haifa, Israel, February 8, 9, 1984, Collection of Papers
Conference supported by Technion - Israel Institute of Technology, Tel Aviv University, Ministry of Defence of Israel, et al. Haifa, Israel, Technion - Israel Institute of Technology, 1984, 358 p. For individual items see A85-37177 to A85-37213. 840000 p. 358 In: EN (English) p.2447

The experimental determination of the effect of nonlinear stiffness on the vibration of elastic structures is considered along with an unsteady wake model of the aerodynamic behavior of a rotor in forward flight, the influence of fighter aircraft load spectrum variations on fatigue crack initiation and growth, a fatigue life evaluation program for the Kfir aircraft, computer-aided tube routing design in aircraft, and a pursuit evasion game with a limited detection range. Attention is given to a crack growth analysis in multiple load path structure, the crack propagation analysis of longitudinal skin cracks in a pressurized cabin, a supersonic panel method based on the triplet singularity, a method to estimate the service life of a case bonded rocket engine, and computational aerodynamics and supercomputers. Other topics explored are related to the concepts and application of aircraft damage tolerance analysis, the influence of initial imperfections on nonlinear free vibration of elastic bars, and wing optimization and fuselage intergration for future generations of supersonic aircraft.
G.R.

Finite element analysis of an ultralight aircraft
(AA)BAUGHN, T. V.; (AB)PACKMAN, P. F.
(AB)(Southern Methodist University, Dallas, TX)
AIAA PAPER 85-0616 IN: Structures, Structural Dynamics, and Materials Conference, 26th, Orlando, FL, April 15-17, 1985, Technical Papers. Part 1 (A85-30226 13-39). New York, American Institute of Aeronautics and Astronautics, 1985, p. 71-78. 850000 p. 8 In: EN (English) p.0

A finite element analysis was conducted to determine the structural integrity of a high wing cable supported ultralight aircraft. A simple symmetrical half structure, macro model was analyzed, subjected to level flight loading, and two wheel landing loading conditions. Flexural and bending stiffness for the supported and unsupported wing were also determined. A preliminary damage tolerance analysis was conducted in which selected cable elements and wing compression struts were removed, and the redistributed loads calculated and possible aircraft flight configurations examined. The model can generate all cable loads, displacement of each structural node, for each loading condition, generate displacement plots, and locate potential highly stressed regions.
Author

Design methods and technology of transport aircraft of today and tomorrow
Methodes de conception et technologie des avions de transport aujourd'hui et demain
(AA)LENSEIGNE, C.
(AA)(Aerospatiale, Toulouse, France)
L'Aeronautique et l'Astronautique (ISSN 0001-9275), no. 107, 1984, p. 4-12. In French. 840000 p. 9 In: FR (French) p.0

The design tools and technology necessary to achieve maximum economy in the aircraft of today and tomorrow are discussed. The objectives of aerodynamic research on transport aircraft are to reduce drag and thus save on fuel, to increase the maximum lifting capacity, and to develop antiturbulence and antiflutter systems. An optimization study of the aircraft structures which is based on the finite element method and an examination of the materials used in the structures is presented. An ergonomic study of aircrew stations reveals the importance of the man-machine interface. Automated active control with applications in reduced longitudinal stability, in airfoil loading control, and in flutter control is considered. The revival of the propellar in short-range aircraft is also discussed.
M.D.

Bonded repairs to surface flaws (by adhesive BFRP patching)
(AA)JONES, R.; (AB)CALLINAN, R. J.
(AB)(Defence Science and Technology Organisation, Aeronautical Research Laboratories, Melbourne, Australia)
Theoretical and Applied Fracture Mechanics (ISSN 0167-8442), vol. 2, Oct. 1984, p. 17-25. 841000 p. 9 refs 8 In: EN (English) p.0

A method using BFRP (boron fiber reinforced plastic) patches for the repair of surface flaws in aluminum alloy aircraft components is presented. The method is best suited to cases where the cracking is primarily due to the presence of inclusions and the stress field is relatively low. A new procedure for repairing cracked bolt holes which involves the use of a bonded insert is also proposed. By using a bonded sleeve, significant reductions in the fretting at the hole and in the stress intensity factors along the crack front have been obtained. The design of preventative repair schemes is illustrated by considering a recent repair to a fairing attachment hole. Both methods, which involve the use of adhesive bonding, have been found to lead to increases in fatigue life.
M.D.

Structural crashworthiness; International Symposium, 1st, University of Liverpool, Liverpool, England, September 14-16, 1983, Invited Lectures
(AA)JONES, N.; (AB)WIERZBICKI, T.
(AA)ED.; (AB)ED.
(AA)(Liverpool, University, Liverpool, England); (AB)(Polska Akademia Nauk, Warsaw, Poland; MIT, Cambridge, MA)
London, Butterworths, 1983, 463 p. No individual items are abstracted in this volume. 830000 p. 463 In: EN (English) p.0

The application of solid, structural, and experimental mechanics to predict the crumpling behavior and energy absorption of thin-walled structures under quasi-static compression and various dynamic crash loadings is examined in reviews of current research. Both fundamental aspects and specific problems in the design of crashworthy aircraft, automobiles, railroad cars, ships, and offshore installations are considered. Topics discussed include laterally compressed metal tubes as impact-energy absorbers, crushing behavior of plate intersections, axial crushing of fiber-reinforced composite tubes, finite-element analysis of structural crashworthiness in the automotive and aerospace industries, crash behavior of aircraft fuselage structures, aircraft crash analysis, ship collisions, and structural damage in airship and rolling-stock collisions. Photographs, graphs, drawings, and diagrams are provided.
T.K.

A method for calculating stress intensity factors in structural elements with curvilinear cracks
Metod rascheta koeffitsientov intensivnosti napriazhenii v elementakh konstruktsii s krivolineinymi treshchinami
(AA)GRISHIN, V. I.; (AB)DONCHENKO, V. IU.
TsAGI, Uchenye Zapiski (ISSN 0321-3429), vol. 14, no. 2, 1983, p. 105-112. In Russian. 830000 p. 8 refs 8 In: RU (Russian) p.3261

An energy method has been developed for determining stress intensity factors in thin-walled structures with multiple-nucleus curvilinear cracks. The procedure has been implemented in a set of specialized software, FITTING, designed for the local strength analysis of aircraft structures. The accuracy of the procedure is evaluated, and examples of stress intensity and crack path computations are presented.
V.L.

Aircraft structures (2nd edition) (Book)
(AA)AZAR, J. J.; (AB)PEERY, D. J.
(AA)(Tulsa, University, Tulsa, OK)
New York, McGraw-Hill Book Co., 1982, 463 p. 820000 p. 463 refs 104 In: EN (English) $37.50 p.2283

An exposition is made of the fundamental concepts in the design and analysis of flight structures, and unified analytical tools are developed for the prediction and assessment of structural behavior irrespective of the field of application. Attention is first given to the definition of a structural system and its constituents, loads, supports, and reactions, as well as to the concepts of statics and the principles of mechanics. There follows a discussion of the basic elasticity relations and of material behavior and selection. The load analysis of flight vehicles and the analysis and design of specific flight vehicle structural components is undertaken using finite difference, stiffness matrix and energy methods for the deflections of structures.
O.C.

Structures, Structural Dynamics and Materials Conference, 25th, Palm Springs, CA, May 14-16, 1984, Technical Papers. Part 1
Conference sponsored by AIAA, ASME, ASCE, and AHS. New York, American Institute of Aeronautics and Astronautics, 1984, 548 p. 840000 p. 548 In: EN (English) p.1908

Papers are presented on topics including optimum reliability-based design of plastic structures; shape optimization; expert systems and computer-aided engineering; stochastic differential equations for structural dynamics; and noise transmission through aircraft panels. Other topics include bolted joints in laminated composites; vacuum degassing behavior of beryllium; the role of ply buckling in the compressive failure of graphite/epoxy tubes; and Langrange shell elements with spurious mode control. For individual items see A84-31627 to A84-31682
J.N.

The problem of the analytical formulation of aircraft surfaces
K zadache analiticheskogo postroeniia poverkhnostei letatel'nykh apparatov
(AA)SNIGIREV, V. F.
Aviatsionnaia Tekhnika (ISSN 0579-2975), no. 4, 1983, p. 100-102. In Russian. 830000 p. 3 refs 5 In: RU (Russian) p.1957

Reference is made to studies by Zav'ialov et al. (1980) and Ahlberg et al. (1972) in which two-dimensional splines were constructed for rectangular regions; one-dimensional splines were set up with respect to coordinate lines. For the case of an arbitrary arrangement of interpolation knots, however, a method of this type cannot be used. In the study made by Marchuk (1977), a two-dimensional spline was set up from the minimization of a certain functional. For this functional, however, the selection of joint approximating functions on the interpolation network of knots involves considerable difficulty. Consideration is therefore given to a different functional, one which, it is shown, makes it easier to find a solution.
C.R.

Status of new aerothermodynamic analysis tool for high-temperature resistant transparencies
(AA)VARNER, M. O.; (AB)BABISH, C. A.
Sverdrup Technology, Inc., Arnold Air Force Station, Tenn. (S6005421)
AD-P003236 F33615-81-C-3412 In Dayton Univ. Conf. on Aerospace Transparent Mater. and Enclosures p 1132-1157 (SEE N84-26596 17-01) 831200 p. 26 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2598

This paper summarizes the status of the definition, selection, modification, and development of a Specific Thermal Analyzer Program for Aircraft Transparencies (STAPAT). The code developed will merge state-of-the-art technology, code accuracy requirements, and the definition of code function requirements resulting in an aerothermodynamic analytical technique that is specifically applicable and limited to the study of high-temperature resistant transparencies for high-speed aircraft. The aerothermodynamic methodologies required for the definition of the convective heat-load requirements of the STAPAT are described. These include the identification of inviscid methodologies covering the subsonic-to-supersonic-speed flight regime complex three-dimensional configurations consisting of real canopy geometries. The external forced convection methodology is described which includes complex three-dimensional effects resulting from the circumferential and streamwise variation of the local heating loads, the forced convection heat transfer as influenced by wall-temperature effects and transition location, and variable-edge entropy effects.
GRA

Aeroelasticity and optimization at the design stage
AEROELASTICITE ET OPTIMISATION EN AVANT-PROJET
(AA)PETIAU, C.; (AB)BOUTIN, D.
Avions Marcel Dassault-Breguet Aviation, Saint-Cloud (France). (A9987704)
In AGARD Aeroelastic Considerations in the Preliminary Design of Aircraft 18 p (SEE N84-11116 02-01) 830900 p. 18 refs 0 In: FR (French) Avail.: NTIS HC A14/MF A01 p.172

A procedure developed for static and dynamic structural analysis is completed in several weeks using a three view scheme of the aircraft, laws of the relative mass of wing units, a summary definition of the internal architecture, a choice of materials, and construction technology. For each version studied, calculation at the design stage involves: (1) a first finite element analysis of rough planning with sampling and simplified loads; (2) analysis of problems of static aeroelasticity, computation of loads by accounting for aeroelasticity, and the automatic study of surrounding loads; (3) computation of flutter with a study of critical configurations with exterior loads removed; and (4) the automatic optimization of sampling, supplying the minimal weight of the structure that satisfies the constraints of static behavior, of aerodistortion limitations, and of the speed of velocity of flutter. A combat aircraft with wings made composite materials is analyzed.
A.R.H.

Validation of the MAGNA (Materially and Geometrically Nonlinear Analysis) computer program for nonlinear finite element analysis of aircraft transparency bird impact
(AA)MCCARTY, R. E.; (AB)HART, J. L.
Air Force Wright Aeronautical Labs., Wright-Patterson AFB, Ohio. (AJ840964)
AD-P003229 In Dayton Univ. Conf. on Aerospace Transparent Mater. and Enclosures p 921-972 (SEE N84-26596 17-01) 831200 p. 52 In: EN (English) Avail.: NTIS HC A08/MF A01 p.2596

The approach taken for validation of MAGNA is based on the simulation of full scale bird impact tests followed by a comparison of the experimental data with that computed by MAGNA. To date, five of these validation studies have been accomplished and several more remain to be conducted. This paper summarizes the results of the validation studies which have been completed to date and lists the user guidelines which have been established in the process. These first validation studies may be characterized as analyses of simple structures, i.e., only single transparent panels have been analyzed as opposed to complex systems of multiple panels joined by metallic edgemember support structure. These same studies may be further characterized as involving only simple definitions of boundary conditions and a somewhat arbitrary procedure for the explicit definition of bird impact pressure loading on the surface of the structure. The cases selected for study were a flat, laminated glass windshield panel; a curved, laminated glass windshield panel; a curved, laminated plastic windshield panel; a bubble-shaped monolithic plastic one-piece canopy; and a heated glass cylinder (which involved neither an aircraft transparency system per se nor bird impact loads).
GRA

Strength-flutter structural optimization of a supersonic cruise combat aircraft
(AA)DOTSON, B. F.
(AA)(Boeing Military Airplane Co., Seattle, WA)
IN: International Symposium on Aeroelasticity, Nuremberg, West Germany, October 5-7, 1981, Collected Papers (A83-49176 24-01). Cologne, Deutsche Gesellschaft fuer Luft- und Raumfahrt, 1982, p. 208-217. 820000 p. 10 refs 12 In: EN (English) p.3547

The computer program FASTOP has been developed by an American aerospace company in concert with the U.S. Air Force Flight Dynamics Laboratory. This program resizes a finite element model of a composite structure to a fully stressed design, provides flutter optimization for a specified flutter speed, and then iterates between strength and flutter for an optimized design. The present investigation is concerned with work which was performed to evaluate the use of the program FASTOP in the aircraft design process, taking into account the difficulties which can arise when it is attempted to optimize real life structures. The approach used in the evaluation of FASTOP involved the application of the program to a supersonic cruise combat aircraft.
G.R.

Algorithmic mass-factoring of finite element model analyses
(AA)PINCHA, P. J.
(AA)(Boeing Military Airplane Co., Seattle, WA)
SAWE PAPER 1451 Society of Allied Weight Engineers, Annual Conference, 41st, San Jose, CA, May 17-19, 1982. 26 p. 820500 p. 26 In: EN (English) p.3009

A serious problem for weight technology engineers is related to the interpretation, manipulation, or conversion of Finite Element Modeled (FEM) Structural Optimization and Analyses Programs (SOAP's) sized structural data into realistic estimates of projected 'as-built' airframe weight. During preliminary design of aircraft structures, particularly low-aspect-ratio wings (having multiloadpaths), initial structural analysis and sizing is almost universally accomplished using a FEM-SOAP system. The output of the FEM-SOAP is the weight of a theoretical structure. However, the weight engineer must use this theoretical weight as a base in the development of an 'as-built' total weight estimate. The present investigation provides a unique algorithmic mass factoring method which accommodates the complexity of the FEM and presents the weight of 'as-built' structure in terms familiar to the weight engineer.
G.R.

Design of an aerobatic aircraft wing using advanced composite materials
(AA)LOUDENSLAGER, L. E.
(AA)(H. D. Neubert and Associates, Inc., Anaheim, CA)
SAE PAPER 821346 Society of Automotive Engineers, Aerospace Congress and Exposition, Anaheim, CA, Oct. 25-28, 1982. 7 p. 821000 p. 7 In: EN (English) p.2463

CAD efforts at tailoring a new configuration for a competition aerobatics aircraft are described. One goal of the privately funded program was to minimize structural weight, thus enhancing the thrust/weight ratio critical to aerobatic maneuvers. The new design featured an optimized tubular steel fuselage, advanced composite wings (graphite epoxy), horizontal and vertical tail, landing gear legs, and secondary structure and fairings. NASA performed computations of the expected aerodynamic loads with a doublet fortex formulation expressed as a two-dimensional inviscid flow finite element model. The NASTRAN program yielded stress contour pilots and the first ten natural frequencies, which were found to reside above the design maneuvering speed due to the use of composite materials.
M.S.K.

Analytical control of the shape of the polygons used in the finite-element method
Analiticheskii kontrol' formy mnogougol'nikov, primeniaemykh v metode konechnykh elementov
(AA)VOROBEV, V. F.
TsAGI, Uchenye Zapiski (ISSN 0321-3429), vol. 13, no. 5, 1982, p. 155-158. In Russian. 820000 p. 4 In: RU (Russian) p.2521

Formulas are derived for analyzing the shape of plane polygonal finite elements specified by nodular point coordinates and element numbers. The analytic procedure proposed here is designed to identify errors in the initial data for a problem solved by means of the finite element method. The procedure also makes it possible to control the shape of the finite elements without using the graphical representation of the computational scheme when solving the problem on a computer. The procedure is demonstrated for a low-aspect-ratio wing.
V.L.

Force method optimization 2. Volume 1: Theoretical development / Final Technical Report, Aug. 1980 - Dec. 1982
(AA)BATT, J. R.; (AB)GELLIN, S.; (AC)GELLATLY, R. A.
Textron Bell Aerospace Co., Buffalo, N. Y. (TV737355) AJ840964
AD-A127073; AFWAL-TR-82-3088-VOL-1 F33615-80-C-3214; AF PROJ. 2307 AFWAL Wright-Patterson AFB, Ohio 821100 p. 126 refs 0 In: EN (English) Avail.: NTIS HC A07/MF A01 p.3358

The document investigates the utilization of the force method of finite element analysis for the automatic iterative design of aircraft structures with stress, displacements, maximum and minimum size and dynamic constraints. It develops a rapid reanalysis method based on the force method for damage assessment. Research has resulted in a computer code named OPTFORCE II an expansion of code OPTFORCE I. Multiple loading capabilities and four finite elements have been included. These are: membrane triangle, membrane quadrilateral, shear panel and bar (axial force). Examples of problems solved by the OPTFORCE II code are presented and compared to the optimization code OPTIM III for purposes of establishing the efficiency of the force method vs. the "displacement' method of analysis. A technical discussion of the research conducted is presented wherein conclusions and recommendations for future research topics are given.
GRA

Development of the advanced composite ground spoiler for C-1 medium transport aircraft
(AA)YAMAUCHI, F.; (AB)MOGAMI, K.; (AC)MASAEDA, H.; (AD)SHIRATA, T.
(AB)(Japan Defense Agency, Technical Research and Development Institute, Tachikawa, Tokyo, Japan); (AD)(Kawasaki Heavy Industries, Ltd., Aircraft Div., Kagamihara, Gifu, Japan)
In: Composite materials: Mechanics, mechanical properties and fabrication; Proceedings of the Japan-U.S. Conference, Tokyo, Japan, January 12-14, 1981. (A82-39851 19-39) Barking, Essex, England, Applied Science Publishers, 1982, p. 504-512. 820000 p. 9 In: EN (English) p.2979

The research and development program for the graphite/epoxy ground spoiler for the C-1 medium transport aircraft is discussed. The design requirement was that the spoiler provide strength and rigidity not less than the baseline spoiler in addition to interchangeability. The design analysis was done by finite element method and the detail design configurations of major structural components were evaluated by trade-off tests in the initial design phase, showing that the components met design requirements. Successful environmental characteristic tests were also conducted. The scattering characteristic of the Gr/E for static and fatigue strength were obtained and found to be significantly superior to that of aluminum alloy. Full scale verification tests were also passed.
C.D.

Finite-element modeling of a fighter aircraft canopy acrylic panel
(AA)LABRA, J. J.
(AA)(Southwest Research Institute, San Antonio, TX)
Journal of Aircraft, vol. 19, June 1982, p. 480-484. 820600 p. 5 In: EN (English) p.2353

A detailed three-dimensional stress analysis of a canopy aircraft acrylic panel was conducted to investigate the probable cause of a recent in-flight acrylic panel failure. The analysis was made using a general-purpose finite-element computer program. Based in part on this analysis, probable design problems associated with the canopy were identified. The study clearly demonstrates that computer-based technology can be successfully used in determining probable causes of failure in geometrically complex structures.
(Author)

Structures and Dynamics Division research and technology plans, FY 1982
(AA)BALES, K. S.
National Aeronautics and Space Administration. Langley Research Center, Hampton, Va. (ND210491)
NASA-TM-84509; NAS 1.15:84509 505-33-33-11 820600 p. 56 In: EN (English) Avail.: NTIS HC A04/MF A01 p.2974

Computational devices to improve efficiency for structural calculations are assessed. The potential of large arrays of microprocessors operating in parallel for finite element analysis is defined, and the impact of specialized computer hardware on static, dynamic, thermal analysis in the optimization of structural analysis and design calculations is determined. General aviation aircraft crashworthiness and occupant survivability is also considered. Mechanics technology required for design coefficient, fault tolerant advanced composite aircraft components subject to combined loads, impact, postbuckling effects and local discontinuities are developed.
S.L.

Finite element thermal analysis of convectively-cooled aircraft structures
(AA)WIETING, A. R.; (AB)THORNTON, E. A.
(AA)(NASA, Langley Research Center, Hampton, VA); (AB)(Old Dominion University, Norfolk, VA)
National Aeronautics and Space Administration. Langley Research Center, Hampton, Va. (ND210491)
In: Numerical methods in heat transfer. (A82-28551 13-34) Chichester, Sussex, England and New York, Wiley-Interscience, 1981, p. 431-443. 810000 p. 13 refs 8 In: EN (English) p.2018

The design complexity and size of convectively-cooled engine and airframe structures for hypersonic transports necessitate the use of large general purpose computer programs for both thermal and structural analyses. Generally thermal analyses are based on the lumped-parameter finite difference technique, and structural analyses are based on the finite element technique. Differences in these techniques make it difficult to achieve an efficient interface. It appears, therefore, desirable to conduct an integrated analysis based on a common technique. A summary is provided of efforts by NASA concerned with the development of an integrated thermal structural analysis capability using the finite element method. Particular attention is given to the development of conduction/forced-convection finite element methodology and applications which illustrate the capabilities of the developed concepts.
G.R.

Conference on Aerospace Transparencies, London, England, September 8-10, 1980, Proceedings
Conference sponsored by the Society of British Aerospace Companies. London, Society of British Aerospace Companies, Ltd., 1981. 713 p (For individual items see A82-24302 to A82-24330) 810000 p. 713 In: EN (English) $59.50 p.1543

Among the aircraft transparency design, testing and analysis topics covered are: (1) transparency development needs for military aircraft in the 1980s, (2) an aircraft transparency design guide, (3) deficiencies and constraints affecting the design of cockpit transparencies and enclosures, (4) bird strikes, (5) windshield system structural enhancement, (6) aircraft transparency bird impact analysis using the MAGNA computer program, (7) stretched acrylic transparency materials, (8) transport aircraft transparencies, and (9) impact resistance test methods. Also considered are (10) abrasion-resistant coatings for aircraft, (11) the role of finite element analysis in the design of birdstrike-resistant transparencies, and (12) the effects of bird orientation on load profile and damage level.
O.C.

Advanced concepts for composite structure joints and attachment fittings. Volume 1: Design and evaluation / Final Report, Jul. 1977 - Dec. 1980
(AA)ALEXANDER, J. V.; (AB)MESSINGER, R. H.
Hughes Aircraft Co., Culver City, Calif. (H4173732) A2024546
AD-A110212; HH-80-402-VOL-1; USAAVRADCOM-TR-81-D-21A DAAJ02-77-C-0-076; DA PROJ. 1L2-62209-AH-76 811100 p. 127 refs 0 In: EN (English) Avail.: NTIS HC A07/MF A01 p.1620

The purpose of this program was to develop the technology of applying fiber-reinforced composite materials to helicopter joints and attachment fittings that permit disassembly of major components. A generic design methodology approach was used to make the data developed applicable to ongoing and future helicopter programs. A detail design, analysis, and testing program was carried out on the three joint and fitting concepts selected: wrapped tension fittings, gearbox attachment fittings, and seat attachment fittings. The scope of the study included analytical design tools, including finite element computer analysis; fabrication techniques, with special emphasis on weight and cost effectiveness considerations; structural integrity testing, including static, dynamic, failsafe/safe-life, and ballistic tolerance considerations; and nondestructive inspection (NDI) techniques.
Author (GRA)

Interpretation and construction of a dynamic similarity model of the A 310 wings
auslegung und bau eines dynamisch aehnlichen modells des a 310 fluegels
(AA)HOENLINGER, H.
Messerschmitt-Boelkow-Blohm G.m.b.H., Ottobrunn (West Germany). (MT620643) Unternehmensbereich Flugzeuge.
MBB-FE-17/S/PUB/42; DGLR-81-035 810505 p. 34 refs 0 Presented at DGLR-Jahrestagung 1981, Aachen, 11-14 May 1981 In: GM (German) Avail.: NTIS HC A03/MF A01 p.1181

The design and construction of a model for directional oscillation flutter for the A-310 wing with supercritical profile is discussed. The rigidity distribution in the model is simulated by a simple metal box. The profile geometry and the mass distribution of the model is copied exactly by a foam structure coated with a fiberglass laminate. A finite element model is used to explain the complicated engine mount, which is supported by a profile beam. Data on rigidity and vibration measurements are included.
Transl. by E.A.K.

A study of the techniques of dynamic analysis of helicopter type structures
(AA)VENN, G. M.; (AB)BOON, D. J.
Westland Helicopters Ltd., Yeovil (England). (WW895582) Dynamics Dept.
In DGLR Seventh European Rotorcraft and Powered Lift Aircraft Forum 21 p (SEE N82-18119 09-01) 810000 p. 21 In: EN (English) Avail.: NTIS HC A99/MF A01 p.1168

The techniques used in the dynamic analysis of helicopter type structures using finite elements are discussed. A test structure was designed and built which incorporated many design features found in helicopter structures. Experiments were performed on this structure to determine the natural frequencies and normal modes. These experimental results were compared with theoretical finite element results. Altering the build state added one or two extra problems so that these could be studied. The modelling problems investigated were: riveted panel-stringer constructions, deep fabricated beams, discontinuous load paths, bolted joints, honeycomb panels, gearboxes, engine mounts, and engines. The techniques used to model these features are discussed. Problems were found when attempting to compare the theoretical and experimental results.
N.W.

Analysis of multiple load path panels containing impact damage
Advisory Group for Aerospace Research and Development, Neuilly-Sur-Seine (France). (AD455458)
In its Design Manual for Impact Damage Tolerant Aircraft Struct. p 195-209 (SEE N82-17160 08-05) 811000 p. 15 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1029

Many structural configurations cannot be represented as monolithic panels in assessing strength degradation from impact damage. The wings of transport aircraft, for example, often consist of skin with riveted stiffeners. The stiffeners can provide damage containment or crack assessment capability that is not considered in element residual strength analysis. Since the crack arrestment capability can significantly improve the residual strength of damaged structure, the stiffening must be included in the analysis. The response of stiffened panels to projectile damage, and available analytical techniques for residual strength prediction, are discussed.
T.M.

Stress concentration in elements of aircraft structures (Russian book)
Kontsentratsiia napriazhenii v elementakh aviatsionnykh konstruktsii
(AA)MAVLIUTOV, R. R.
Moscow, Izdatel'stvo Nauka, 1981. 144 p. In Russian. 810000 p. 144 refs 216 In: RU (Russian) p.4125

The book deals with methods of calculation and experimental determination of stress concentrations in typical elements of aircraft structures. Emphasis is placed on problems of stress concentration in parts subjected to elastic, plastic, and creep deformation under complex loading conditions. The effect of loading history is evaluated. Computer algorithms and programs for stress calculation are presented together with design optimization guidelines.
V.L.

Adaptive finite element technology in integrated design and analysis / Final Report, 1 Oct. 1979 - 31 Dec. 1980 (aircraft structures design)
(AA)SZABO, B. A.; (AB)BASU, P. K.; (AC)DUNAVANT, D. A.; (AD)VASILOPOULOS, D.
Washington Univ., St. Louis, Mo. (WG032961) Center for Computational Mechanics.
NASA-CR-164560; WU/CCM-81/1 NSG-1640 810100 p. 94 refs 0 In: EN (English) Avail.: NTIS HC A05/MF A01 p.2682

An assessment of the potential impact of adaptive finite element technology on the analysis part of the aircraft structural synthesis process is presented. The main conclusion is that adaptive application of the p-version of the finite element method based on indirect error estimation procedures results in substantial cost reduction and increased reliability of the computed data. Adaptivity based on direct a posteriori error estimation has the potential for additional savings.
M.G.

Use of optimization in helicopter vibration control by structural modification
(AA)DONE, G. T. S.; (AB)RANGACHARYULU, M. A. V.
(AA)(City University, London, England); (AB)(Birla Institute of Technology and Science, Pilani, India)
(European Rotorcraft and Powered Lift Aircraft Forum, European Rotorcraft and Powered Lift Aircraft Forum, 5th, Amsterdam, Netherlands, Sept. 4-7, 1979.) Journal of Sound and Vibration, vol. 74, Feb. 22, 1981, p. 507-518. 810222 p. 12 refs 9 In: EN (English) p.1580

The application of optimization methods to helicopter crew area vibration reduction by means of structural modification is studied. With stiffness parameters as design variables, forced vibration response circles are used to identify the parameters most effective in controlling crew area response, thereby reducing them. The problem is cast as a nonlinear programming problem, and a sequential unconstrained minimization technique incorporating an algorithm is used to determine the precise values of the parameters. Although too simple for actual engineering design use, the model demonstrates what optimization routines make possible.
O.C.

An aeronautical structural analysis system for static analysis /HAJIF-I/
(AA)FENG, Z.
(AA)(Chinese Aeronautical Establishment, Communist China)
Acta Aeronautica et Astronautica Sinica, vol. 1, no. 1, 1980, p. 16-26. In Chinese, with abstract in English. 800000 p. 11 In: CH (Chinese) p.1440

The HAJIF-I aeronautical structural analysis system for static analysis is the first large software system developed by the Chinese Aeronautical Establishment. By using multilevel substructure analysis and the finite element displacement method, this system is suitable for various linear static analyses of aeronautical structures. The maximum capacity of the system is 3000 nodal DOF of each substructure, 99 substructures, and 10 levels of substructures. The system provides some statements for automatic data generation; and there are several special statements for structural analysis, enabling the user to organize his own computation flow. A user's specification for stress analysis of an aircraft is presented as an example.
B.J.

A conversational, topological grid method and optimization of structural calculations involving finite elements
maillage par methode topologique conversationnelle et optimisation dans les calculs de structure par elements finis
(AA)PETIAU, C.
Avions Marcel Dassault-Breguet Aviation, Saint-Cloud (France). (A9987704)
AAAF-NT-79-30; ISBN-2-7170-0578-I Association Aeronautique et Astronautique de France Paris 790000 p. 32 refs 0 Presented at 14th Intern. AAAF Aeron. Congr., Paris, 6-8 Jun. 1979 In: FR (French) Avail.: NTIS HC A03/MF A01; CEDOCAR, Paris FF 34 (France and EEC) FF 39 (others) p.1003

A finite element code used in the structural analysis and design of MIRAGE aircraft is studied. Particular attention is given to two modules of this program: (1) the elaboration of a three dimensional grid representation, based on a topological method; and (2) a linear optimization of the structural stability parameters. A progressive shift in data input techniques towards a man machine conversational mode is also discussed. Stress concentration results, including aeroelasticity data, and associated computer graphics are shown, using the MIRAGE 2000 aircraft as an example.
Author (ESA)

Sonic fatigue design techniques for advanced composites aircraft structures / Final Technical Report, Aug. 1977 - Dec. 1979
(AA)HOLEHOUSE, I.
Rohr Industries, Inc., Chula Vista, Calif. (RY945193) AJ840964
AD-A090553; RHR-80-019; AFWAL-TR-80-3019 F33615-77-C-3033 800400 p. 343 refs 0 In: EN (English) Avail.: NTIS HC A15/MF A01 p.301

A combined analytical and experimental program was conducted in order to develop a semi-empirical sonic fatigue design method for curved and flat graphite-epoxy skin-stringer panels. A range of multi-bay panels was subjected to high intensity noise environments in a progressive wave tube. Shaker tests were also performed in order to provide additional random fatigue data. Finite-element analyses were carried out on the test panel designs, generating static strains and frequencies. Multiple stepwise regression analysis was used to develop the sonic fatigue design method. Design equations and a nomograph are presented. Comparisons of sonic fatigue resistance between graphite and aluminum panels were also carried out. The design method developed is presented as a self-contained section in this report and is suitable for practical design use.
GRA

Green's functions for stresses, stress intensity factors, and displacements in a cracked, infinite, isotropic sheet under symmetric loads
(AA)RODERICK, G. L.
(AA)(U.S. Army, Structures Laboratory, Hampton, Va.)
Engineering Fracture Mechanics, vol. 13, no. 1, 1980, p. 95-105. 800000 p. 11 refs 9 In: EN (English) p.2358

Green's functions for stresses, stress intensities, and displacements were derived for an infinite cracked isotropic sheet under point symmetric loading. First, complex stress functions were derived for four point symmetric concentrated loads acting on a cracked sheet. Then, the functions and their appropriate derivatives were used to express stresses, stress intensities, and displacements in terms of unit load components. Stresses and displacements calculated by use of the Green's functions were compared with coarse-grid finite-element calculations, primarily as a test for the existence of algebraic errors. The calculations were in good agreement. The results should be useful in the analysis and design of damage-tolerant aircraft structures.
(Author)

A study of the reinforcement required for cutouts in aircraft semi-monocoque structure / Ph.D. Thesis
(AA)MOTLEY, G. R.
Southern Methodist Univ., Dallas, Tex. (SS949356)
800000 p. 280 In: EN (English) Avail: Univ. Microfilms Order No. 8012578 p.2084

An automated solution procedure was developed which results in substantial savings in time and cost required for static, ultimate strength analysis of cutouts in aircraft monocoque structures. A specialized preprocessor was developed to use with existing software which automatically generates the bulk of the input data. A postprocessor was also developed which evaluates the results, revises member properties, and iterates until a near optimal solution is obtained. A fatigue analysis is also required for most cutout installations. Data to support fatigue analysis were extended by use of finite element methods to cover most common cutout reinforcement configurations. General fatigue design guidelines available in the literature were summarized. A third item to consider in development of any cutout reinforcement is the method of fabrication to be used for individual elements. Decisions should be based on relative costs of the various techniques available. A series of studies based in part on the cost of an actual cutout installation were completed and the results summarized to support future design efforts.
Dissert. Abstr.

Composite structural materials / Semiannual Report, Apr. - Sep. 1979
(AA)ANSELL, G. S.; (AB)LOEWY, R. G.; (AC)WIBERLEY, S. E.
Rensselaer Polytechnic Inst., Troy, N. Y. (RO935231)
NASA-CR-162578; SAR-37 NGL-33-018-003 Sponsored in part by AFOSR 791200 p. 107 refs 0 In: EN (English) Avail.: NTIS HC A06/MF A01 p.695

A multifaceted program is described in which aeronautical, mechanical, and materials engineers interact to develop composite aircraft structures. Topics covered include: (1) the design of an advanced composite elevator and a proposed spar and rib assembly; (2) optimizing fiber orientation in the vicinity of heavily loaded joints; (3) failure mechanisms and delamination; (4) the construction of an ultralight sailplane; (5) computer-aided design; finite element analysis programs, preprocessor development, and array preprocessor for SPAR; (6) advanced analysis methods for composite structures; (7) ultrasonic nondestructive testing; (8) physical properties of epoxy resins and composites; (9) fatigue in composite materials, and (10) transverse thermal expansion of carbon/epoxy composites.
A.R.H.

Computer analysis of semi-monocoque shell sections
(AA)SHEPS, Z.; (AB)RAIBSTONE, A. I.; (AC)BARUCH, M.
(AB)(Israel Aircraft Industries, Ltd., Lod, Israel); (AC)(Technion - Israel Institute of Technology, Haifa, Israel)
Computers and Structures, vol. 9, Sept. 1978, p. 305-313. 780900 p. 9 In: EN (English) p.4361

A computer program for the analysis of semimonocoque structures loaded through a rigid bulkhead is presented. The program computes the structure's principal moments of inertia, principal axes, shear center, and torsional rigidity as well as its shear flows and inertial axial loads. Furthermore, the program can convert forces and moments at one of the structure ends into a system of statically equivalent forces acting on the grid points of this region. This option is essential when a structural analysis is continued utilizing the finite methods, and the boundary conditions of the structure must be accurately defined.
V.T.

Recent developments at ONERA in the field of structural analysis methods
Recents developpements a l'ONERA dans les methodes de calcul des structures
(AA)VALID, R.
ONERA, TP NO. 1979-79 (Congres International d'Aeronautique, 14th, Paris, France, June 6-8, 1979.) ONERA, TP no. 1979-79, 1979. 33 p. In French. 790000 p. 33 refs 38 In: FR (French) p.4168

Attention is given to such developments at ONERA as variational principles for finite element methods, research on the application of various methods (i.e., substructuring, perturbation, Ritz methods, various algorithms, etc.) which aim to decrease computational cost, with increase of accuracy, and research on the homogenization of materials or structures with periodic characteristics, composite materials, and model identification. Attention is also given to research on transient strain response computation, dynamic response with unilateral contact, and buckling problems. Applications include hydroelasticity and elastoacoustics.
B.J.

Prediction of the angular response power spectral density of aircraft structures / Final Report, Jan. 1976 - Jun. 1978
(AA)LEE, J. H.; (AB)OBAL, M., W.; (AC)BROWN, D. L.
Air Force Flight Dynamics Lab., Wright-Patterson AFB, Ohio. (AI058438)
AD-A066141; AFFDL-TR-78-188 781200 p. 98 refs 0 In: EN (English)
Avail.: NTIS HC A05/MF A01 p.1932

The design of airborne electro-optical systems requires the knowledge of angular vibration as well as linear vibration of aircraft structures. Rather than predicting the angular vibration subject to aerodynamic and acoustic excitations, an attempt is made here to relate the angular vibration directly to the linear vibration response. With the Bernoulli-Euler beam used as a theoretical model, a relationship has been derived between the linear and angular vibration power spectral density functions. Based on this relationship together with the angular root-mean-squared vibration amplitude as previously predicted by Lee and Whaley (AFFDL-TR-76-56, AF Flight Dynamics Laboratory, Wright-Patterson AFB, Ohio), it is now possible to predict the angular power spectral density and a length scale associated with the angular measurement technique. Tested on the typical flight test data of RF-4C and F-15 fighters, CH-3E helicopter, and B-52 bomber, the predicted angular power spectral density lies within a + or - 0 db band about the measurement. Though crude, such a prediction is useful in the preliminary design stage in that one can quickly estimate the angular vibration environment prior to fabrication.
Author (GRA)

Adaptive approximations in finite element structural analysis (for aircraft components)
(AA)PEANO, A.; (AB)PASINI, A.; (AC)RICCIONI, R.; (AD)SARDELLA, L.
(AA)(Milano, Politecnico, Milan, Italy); (AD)(Istituto Sperimentale Modelli e Strutture, Bergamo, Italy)
In: Trends in computerized structural analysis and synthesis; Proceedings of the Symposium, Washington, D.C., October 30-November 1, 1978. (A79-22926 08-39) Oxford and Elmsford, N.Y., Pergamon Press, 1978, p. 333-342. 780000 p. 10 refs 19 In: EN (English) p.1420

A finite element computer program is termed adaptive when it possesses a local a posteriori error estimation capability, along with a capability of assigning (automatically or with minimum user interaction) additional degrees of freedom to regions with particularly high accuracy requirements. The automated convergence process reduces the discretization error until the accuracy desired is obtained, thereby establishing confidence in the solution. The purpose of the present paper is to demonstrate, on the basis of two- and three-dimensional applications, an adaptive computer program capable of obtaining improved solutions at minimum cost. The behavior of p-convergent approximations at crack tip singularities is studied, and the implementation of adaptive finite element techniques into second generation large-scale computer programs is discussed.
V.P.

Analysis of aircraft structure using applied fracture mechanics
(AA)WILHEM, D. P.
Northrop Corp., Hawthorne, Calif. (N5631231) Aircraft Group.
In AGARD Fracture Mech. Design Methodology 17 p (SEE N79-20409 11-39) 790100 p. 17 refs 0 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1437

An aircraft designed and analyzed for a particular set of usages is often placed in a service environment which is more severe than originally planned. The consequence of this occurrence is that many design details such as cutouts, holes, etc., are placed in a spectrum of loads which result in higher operating stresses. In the original full scale fatigue test, a different (design usage) spectrum is usually employed, and can only indicate fatigue critical areas. Using the finite element approach with stress intensity values and usage spectra, estimates are made of the crack growth life for a part-through-crack at a cutout. These data are then used to establish inspection intervals. Three distinct spectra were developed to represent usage, and analytical/experimental correlation was made for those spectra. In the majority of cases, good agreement was obtained. For these cases where the correlation is not good, refinements need to be made to the stress intensity solutions and/or the crack growth model. The reliance on more than one method of analysis is recommended for stress intensity evaluation of fatigue and fracture-critical areas. A comparison of the methods used in determining crack growth parameters sometimes indicates that the added cost of a more complex technique is not warranted, particularly when parametric design studies are involved. The use of a newer approach to the prediction of both fatigue crack growth and residual strength, employing a wide range resistance curve, is promising. Its usefulness in pinpointing differences in the cutout problem is given.
A.R.H.

Cyclic linkage of finite elements with application (to aircraft structural analysis)
(AA)ATANACKOVIC, B.
(AA)(Aerotechnical Institute, Belgrade, Yugoslavia)
IAF PAPER 78-213 International Astronautical Federation, International Astronautical Congress, 29th, Dubrovnik, Yugoslavia, Oct. 1-8, 1978, 20 p. 781000 p. 20 refs 8 In: EN (English) p.98

Two models for linking finite elements in the analysis of aircraft structures are proposed in which a polygonal element is the basic carrier of information on the structure. Information on the structure is obtained by optimal organization of the elements. Cyclic linkage of the polygonal elements is used to model two-dimensional geometric aircraft structures; it also provides graphical representation of stress and geometric data. The method is illustrated by examples.
V.P.

Stress intensity analysis: Analytical, finite element for surface flaws, holes
(AA)WILHEM, D. P.
Northrop Corp., Hawthorne, Calif. (N5631231) Aircraft Group.
In AGARD Fracture Mech. Design Methodology 19 p (SEE N79-20409 11-39) 790100 p. 19 refs 0 In: EN (English) Avail.: NTIS HC A11/MF A01 p.1436

Several methods are available to obtain stress intensity for developing cracks in structure where uniform loading and symmetric cracks prevail. Unfortunately in all aircraft structure both loading (stress) and crack geometries are far from ideal. These factors combined with localized plasticity require the use of more sophisticated means of obtaining stress intensity factors. Finite element analysis, both with and without special cracked elements, can be used to obtain stress intensity values. Careful attention must be paid in modeling to account for various factors, i.e., fasteners, etc., which affect the stress field. In many cases where elastic-plastic behavior is evident, those finite element programs with nonlinear capability can be effectively used to compute J-integral values for use in both fatigue and fracture studies. One case study presented involves a cutout in the wing in a highly stressed region the root. Other cases deal with part-through-cracks at holes and countersinks and other design details. The use of three dimensional finite element models to obtain stress intensities for cracks at holes provides an opportunity to evaluate the merits of each method of analysis; analytical, finite element and semi-empirical. Comparisons are presented for several cases.
A.R.H.

Fast algorithm for calculating a family of wing structures
Bystrodeistvuiushchii algoritm rascheta odnogo semeistva kryl'evykh konstruktsii
(AA)KULCHENKO, G. G.; (AB)PERVAK, V. D.; (AC)RIABCHENKO, V. M.; (AD)STOPKEVICH, V. G.
Samoletostroenie - Tekhnika Vozdushnogo Flota, no. 42, 1977, p. 68-73. In Russian. 770000 p. 6 refs 6 In: RU (Russian) p.999

A fast finite-element algorithm is proposed for calculating the aerodynamic forces (horizontal, vertical and shear) on a wing of longeron-rib configuration. The computation is performed in three stages: (1) the static stress state is constructed on the basis of the theory of thin rods, (2) self-equilibrated stress states, localized on small sections of the wing, are chosen as extra unknowns, and (3) matrix coefficients are claculated. The method developed here can be used to calculate the wing as a whole, as well as the separate substructures of a complex wing.
B.J.

Recent developments in analysis of crack propagation and fracture of practical materials (stress analysis in aircraft structures)
(AA)HARDRATH, H. F.; (AB)NEWMAN, J. C., JR.; (AC)ELBER, W.; (AD)POE, C. C., JR.
National Aeronautics and Space Administration. Langley Research Center, Hampton, Va. (ND210491)
NASA-TM-78766 505-02-33-03 780600 p. 20 refs 0 Presented at the Intern. Symp. of Fracture Mechanics, Washington, D. C., 11-13 Sep. 1978 In: EN (English) Avail.: NTIS HC A02/MF A01 p.2828

The limitations of linear elastic fracture mechanics in aircraft design and in the study of fatigue crack propagation in aircraft structures are discussed. NASA-Langley research to extend the capabilities of fracture mechanics to predict the maximum load that can be carried by a cracked part and to deal with aircraft design problems are reported. Achievements include: (1) improved stress intensity solutions for laboratory specimens; (2) fracture criterion for practical materials; (3) crack propagation predictions that account for mean stress and high maximum stress effects; (4) crack propagation predictions for variable amplitude loading; and (5) the prediction of crack growth and residual stress in built-up structural assemblies. These capabilities are incorporated into a first generation computerized analysis that allows for damage tolerance and tradeoffs with other disciplines to produce efficient designs that meet current airworthiness requirements.
A.R.H.

Static and dynamic analysis of helicopter structures with the finite element method
Analyse statique et dynamique des structures d'helicoptres par la methode des elements finis
(AA)AUDRY, R.
(AA)(Societe Nationale Industrielle Aerospatiale, Division Helicopteres, Marignane, Bouches-du-Rhone, France)
Vertica, vol. 1, no. 4, 1977, p. 255-262. In French. 770000 p. 8 refs 5 In: FR (French) p.745

General-purpose programs for finite element analysis of the static and dynamic response of helicopters are described. The library of elements for the programs includes longitudinal sections acting in the traction/compression mode, plane surfaces acting as membranes, longitudinal sections capable of bending, plane surfaces capable of bending, and volumes. The dynamic response program is limited primarily by the computing time associated with the rigidity matrix. An application of the finite element programs to the SA 341 Gazelle helicopter is presented.
J.M.B.

Structural wing-fuselage static interaction by a combined method of tests and numerical analyses (for Kfir aircraft)
(AA)KALEV, I.; (AB)BLASS, E.; (AC)BARUCH, M.
(AB)(Technion - Israel Institute of Technology, Haifa, Israel)
Journal of Aircraft, vol. 14, Dec. 1977, p. 1186-1191. 771200 p. 6
In: EN (English) p.539

Structural wing-fuselage interaction has a major influence on the Kfir aircraft stress distributions. A method that combines both numerical computations and sparate full-scale nondestructive static tests of both wing and fuselage is described, and the selection of this approach is substantiated. The method incorporates five main interdependent activities: (1) performance of static tests on a full-scale wing; (2) performance of static tests on a full-scale fuselage; (3) construction of a finite-element model of the wing; (4) construction of finite-element models of the fuselage frames to which the wing is attached; and (5) computation of the wing-fuselage interaction, using the force method and considering the wing and the fuselage as two substructures.
(Author)

AGARD Lecture Series No.147 Advisory Group for Aerospace Research and Development, NATO PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN Published August 1986 140 pages This Lecture Series will introduce the main aspects of finite element analysis and modelling with emphasis placed on the solution of practical design problems. An outline will be given of the broad principles of the finite element method with some emphasis on the limitations of the technique. This will be followed by an explanation of the modelling problems encountered in the analysis of real structures together with their resolution. The use of one of the FE P.T.O	AGARD-LS-147 Aircraft panels Aircraft Mathematical models Models Design Structural analysis

AGARD Lecture Series No.147 Advisory Group for Aerospace Research and Development, NATO PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN Published August 1986 140 pages This Lecture Series will introduce the main aspects of finite element analysis and modelling with emphasis placed on the solution of practical design problems. An outline will be given of the broad principles of the finite element method with some emphasis on the limitations of the technique. This will be followed by an explanation of the modelling problems encountered in the analysis of real structures together with their resolution. The use of one of the FE P.T.O	AGARD-LS-147 Aircraft panels Aircraft Mathematical models Models Design Structural analysis

AGARD Lecture Series No.147 Advisory Group for Aerospace Research and Development, NATO PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN Published August 1986 140 pages This Lecture Series will introduce the main aspects of finite element analysis and modelling with emphasis placed on the solution of practical design problems. An outline will be given of the broad principles of the finite element method with some emphasis on the limitations of the technique. This will be followed by an explanation of the modelling problems encountered in the analysis of real structures together with their resolution. The use of one of the FE P.T.O	AGARD-LS-147 Aircraft panels Aircraft Mathematical models Models Design Structural analysis

AGARD Lecture Series No.147 Advisory Group for Aerospace Research and Development, NATO PRACTICAL APPLICATION OF FINITE ELEMENT ANALYSIS TO AIRCRAFT STRUCTURAL DESIGN Published August 1986 140 pages This Lecture Series will introduce the main aspects of finite element analysis and modelling with emphasis placed on the solution of practical design problems. An outline will be given of the broad principles of the finite element method with some emphasis on the limitations of the technique. This will be followed by an explanation of the modelling problems encountered in the analysis of real structures together with their resolution. The use of one of the FE P.T.O	AGARD-LS-147 Aircraft panels Aircraft Mathematical models Models Design Structural analysis

systems will be included.

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

ISBN 92-835-1536-6

systems will be included.

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

ISBN 92-835-1536-6

systems will be included.

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

ISBN 92-835-1536-6

systems will be included.

The material in this publication was assembled to support a Lecture Series under the sponsorship of the Structures and Materials Panel and the Consultant and Exchange Programme of AGARD presented on 11—12 September 1986 in Geilo, Norway, 15—16 September 1986 in Lisbon, Portugal, 29—30 September 1986 at McClellan AFB, CA, USA, 2—3 October 1986 at Kelly AFB, TX, USA and 6—7 October 1986 at Wright-Patterson AFB, OH, USA.

ISBN 92-835-1536-6

AGARD

NATO OTAN

7 rue Ancelle · 92200 NEUILLY-SUR-SEINE

FRANCE

Telephone (1)47.38.57.00 · Telex 610 176

DISTRIBUTION OF UNCLASSIFIED AGARD PUBLICATIONS

AGARD does NOT hold stocks of AGARD publications at the above address for general distribution. Initial distribution of AGARD publications is made to AGARD Member Nations through the following National Distribution Centres.Further copies are sometimes available from these Centres, but if not may be purchased in Microfiche or Photocopy form from the Purchase Agencies listed below.

NATIONAL DISTRIBUTION CENTRES

BELGIUM
Coordonnateur AGARD — VSL
Etat-Major de la Force Aérienne
Quartier Reine Elisabeth
Rue d'Evere, 1140 Bruxelles

CANADA
Defence Scientific Information Services
Dept of National Defence
Ottawa, Ontario K1A 0K2

DENMARK
Danish Defence Research Board
Ved Idraetsparken 4
2100 Copenhagen Ø

FRANCE
O.N.E.R.A. (Direction)
29 Avenue de la Division Leclerc
92320 Châtillon

GERMANY
Fachinformationszentrum Energie,
Physik, Mathematik GmbH
Kernforschungszentrum
D-7514 Eggenstein-Leopoldshafen

GREECE
Hellenic Air Force General Staff
Research and Development Directorate
Holargos, Athens

ICELAND
Director of Aviation
c/o Flugrad
Reyjavik

ITALY
Aeronautica Militare
Ufficio del Delegato Nazionale all'AGARD
3 Piazzale Adenauer
00144 Roma/EUR

LUXEMBOURG
See Belgium

NETHERLANDS
Netherlands Delegation to AGARD
National Aerospace Laboratory, NLR
P.O. Box 126
2600 AC Delft

NORWAY
Norwegian Defence Research Establishment
Attn: Biblioteket
P.O. Box 25
N-2007 Kjeller

PORTUGAL
Portuguese National Coordinator to AGARD
Gabinete de Estudos e Programas
CLAFA
Base de Alfragide
Alfragide
2700 Amadora

TURKEY
Millî Savunma Başkanlığı (MSB)
ARGE Daire Başkanlığı (ARGE)
Ankara

UNITED KINGDOM
Defence Research Information Centre
Kentigern House
65 Brown Street
Glasgow G2 8EX

UNITED STATES
National Aeronautics and Space Administration (NASA)
Langley Research Center
M/S 180
Hampton, Virginia 23665

THE UNITED STATES NATIONAL DISTRIBUTION CENTRE (NASA) DOES NOT HOLD STOCKS OF AGARD PUBLICATIONS, AND APPLICATIONS FOR COPIES SHOULD BE MADE DIRECT TO THE NATIONAL TECHNICAL INFORMATION SERVICE (NTIS) AT THE ADDRESS BELOW.

PURCHASE AGENCIES

Microfiche or Photocopy
National Technical
Information Service (NTIS)
5285 Port Royal Road
Springfield
Virginia 22161, USA

Microfiche
ESA/Information Retrieval Service
European Space Agency
10, rue Mario Nikis
75015 Paris, France

Microfiche or Photocopy
The British Library
Document Supply Division
Boston Spa, Wetherby
West Yorkshire LS23 7BQ
England

Requests for microfiche or photocopies of AGARD documents should include the AGARD serial number, title, author or editor, and publication date. Requests to NTIS should include the NASA accession report number. Full bibliographical references and abstracts of AGARD publications are given in the following journals:

Scientific and Technical Aerospace Reports (STAR)
published by NASA Scientific and Technical
Information Branch
NASA Headquarters (NIT-40)
Washington D.C. 20546, USA

Government Reports Announcements (GRA)
published by the National Technical
Information Services, Springfield
Virginia 22161, USA

Printed by Specialised Printing Services Limited
40 Chigwell Lane, Loughton, Essex IG10 3TZ

ISBN 92-835-1536-6