Imperial College
of Science Technology and Medicine

DAJA 45-91-M-0227.
R&D 6703-MS-03.

Centre for Composite Materials

WORKSHOP:

FINITE ELEMENT METHODS FOR COMPOSITES

5th & 6th September 1991
Imperial College, London, UK.

Organised by the Centre for Composite Materials,
Imperial College of Science, Technology & Medicine,
Prince Consort Road,
London, SW7 2BY.

Supported by the European Research Office, United States Army and the European Office of Aerospace Research & Development, United States Air Force, under Contract/Purchase Order No. DAJA 45-91-M-0227.

SUMMARY

This document contains a review of the Workshop proceedings, together with the extended abstracts submitted by the speakers.

CONTENTS

Finite Elements in Composite Materials Workshop

Papers

* Dr. Miravete was unable to present his paper. An additional paper was read in its place by Dr. N. Zahlan, ICI.

INTRODUCTION

The objective of this Workshop was to consider the current position of applying the finite element method to applications involving composite materials. It was expected that issues needing attention would be identified.

To achieve these ends 13 speakers representing several aspects of the field, from a number of countries (France, 1; Germany, 2; Netherlands, 1; Spain, 1; UK, 5; USA, 3) were invited. All accepted, but one withdrew just before the meeting. This talk was replaced by a contribution from one of the delegates. There was one further contributed paper.

Seven software companies initially agreed to participate, via discussions and demonstrations; finally only three attended. Some withdrew because they wanted significantly more time for demonstrating than was scheduled in the programme.

In total 52 persons attended the workshop made up as follows: 13 speakers, 28 delegates, 3 representatives from software suppliers, 8 Imperial College staff (see Appendix C).

The meeting was chaired by Dr. I. C. Taig, ex-British Aerospace, Consultant to NAFEMS (National Agency for Finite Element Methods & Standards).

PROCEEDINGS

The format of the workshop allowed for 10 minutes discussion after each presentation, with additional discussion periods scheduled at the end of the first day and at the end of the meeting. To keep within time constraints discussion always had to be stopped after each paper, although the general discussion periods appeared to be of adequate length to allow all present to make their points.

Extended abstracts of the invited papers and original contributed paper are included as Appendix A. The substituted paper (given subsequently at the ICCS-6 Conference) is included as Appendix B.

In his opening address Dr. Taig noted that none of the currently available finite element systems gave a true representation of a composite. The requirements could be split into four categories (essential, expected, desirable, special) which can then be addressed under a number of areas: materials, geometry, formulation, solution, results, failure [Ref: 'Finite Element Analysis of Composite Materials', I. C. Taig, NAFEMS, East Kilbride, Scotland, 1990].

Many of the speakers addressed most of the points listed by Dr. Taig. In particular the following were emphasised: the need for an accurate description of the material; the need to model the failure process; the need for systems that are easy to use; the fact that different packages can produce different results.

The conclusions from the discussions and presentations are given in the next section.

CONCLUSIONS

The workshop was generally agreed to have been a success and to be worth repeating. The technical conclusions were as follows:

Materials	-	the actual material must be modelled
	-	properties must be appropriate to the scale of the analysis
Geometry	-	the system must allow for correct orientation of the material; independent of the mesh
	-	it should be possible to model realistic lay-ups
	-	it should be possible to represent (ply) discontinuities
Formulation	-	it should be possible to incorporate through-thickness modelling
	-	specific formulation is needed to allow for iteration
Solution	-	stable methods are needed to model damage propagation
	-	it should be possible to include material and geometric non-linearity
Results	-	need transverse and through-thickness (as well as in-plane) results
	-	need flexible post-processing, with ability to discard irrelevant information
Failure	-	include relevant critical properties
	-	system must be able to identify failure mode(s)
	-	must be able to model progression of failure to final collapse
General	-	consider using spreadsheets to interface input data with FE system
	-	users need more, and better, communication with software vendors
	-	establish a system (benchmarks) for verifying results

RECOMMENDATIONS

- the workshop organisers to consider repeating the event in, say, two years' time.
- attendees to be sent a questionnaire by the organisers to seek views on format and content of a future meeting.
- software vendors and NAFEMS to be sent a copy of the current document in order to encourage action on the conclusions of the workshop.

We are all familiar with the potential of both composite materials and finite element analysis. It is difficult to realise the full potential of a composite, and there must be many instances of such materials being abandoned in favour of metals because of a lack of understanding of some fundamental aspect of the composite's performance. Equally, incorrect answers with consequent reflection on the behaviour of the product, can be obtained from a finite element analysis if insufficient attention is paid to details of the modelling.

When it comes to using finite elements with composites we have at our disposal greater potential but more scope for making mistakes. In view of the increasing use of both areas of technology it seemed timely, therefore, to hold a workshop to discuss the issues involved, to identify problem areas, and to recommend ways of improving our use of F-E with these materials.

In putting this meeting together we tried to assemble speakers who are practitioners, and can thus talk about real problems and how they solved them, in several fields of application, together with software suppliers. The workshop will only succeed if we have a dialogue between users and suppliers.

We are extremely grateful for the financial support provided by the European Research Offices of the United States Army & Air Force. We are particularly grateful for the encouragement provided in the early planning stage by Dr. Wilbur Simmons of the US Army Office.

F.L. Mathews,
Director, Imperial College Centre for Composite Materials.

ANALYSIS OF LAMINATED COMPOSITES BEYOND INITIAL FAILURE

WORKSHOP: FINITE ELEMENT METHODS FOR COMPOSITES

Imperial College of Science Technology and Medicine
University of London
September 5-6, 1991

Fu-Kuo Chang

Department of Aeronautics and Astronautics
Stanford University, Stanford, CA 94305

ABSTRACT

In order to analyze the mechanical response of fiber-reinforced laminated composites, an analytical model is required which can calculate accurately stresses, strains and deformations of the composites, and predict reliably the state of damage and the material properties at every damaged state inside the materials as a function of the applied load and/or the history of the load. Due to inhomogenity and anisotrophy, stress distributions inside composites are very complicated and may vary from layer to layer and location to location. As a consequence, damage in composites involves multiple failure modes, which are strongly influenced by stress and strain distributions inside the materials.

Finite element methods have been developed and utilized in the literature for calculating stresses and deformations of laminated composites before the occurrence of damage. Once the materials suffer some degree of damage, an appropriate failure analysis is needed in conjunction with the finite element analysis for analyzing the post-damage response of the composites. In general, failure modes in laminated composites can be classified into two types: in-plane failure such as fiber breakage, fiber comprescssion, matrix cracking, matrix compression, fiber matrix shearing, and out-of-plane failure such as delamination. The failure mechanism of each failure mode is different; hence, the effect of damage on the material properties is also strongly dependent upon the mode of failure.

Several methods have been proposed in the literature for predicting failure of laminated composites beyond initial failure. In this presentation, a progressive failure analysis based on the continuum mechanics concept will be presented for predicting the in-plane damage in composites. The method consists of a nonlinear finite element analysis for calculating stresses, and a failure analysis for predicting the type and the extent of damage in the materials as a function of the applied load. The application of this analysis will be demonstrated through modeling the response of laminated composites containing holes and cutouts.

For analyzing delamination failure, a model will be presented which consists of a nonlinear finite element method for calculating stresses, a contact/slip analysis for modeling the interfacial condition of the delamination surfaces during loading, and a crack growth criterion for predicting delamination propagation. The model will be demonstrated for modeling the delamination propagation in composites resulting from compression or transverse loading. Discussion will also be given on the limitation of the application of both analyses and the current efforts for improving the analyses.

FINITE ELEMENT RESULTS FOR COMPOSITES IN ARMAMENT APPLICATIONS: PRESSURIZED CYLINDERS AND FRACTURE TEST METHODS

by J.H. Underwood and M.D. Witherell

US Army Armament Research, Development and Engineering Center
Benet Laboratories Watervliet, NY 12189 USA

Hollow cylinder applications for composites will be discussed, as related to armament components. Results of recent and ongoing stress analysis of various multiorthotropic-layered cylinders will be described, simulating the type of applied and residual loads common to thick-wall cylinders and the material properties of both metallic and composite materials.

The need for fracture mechanics analysis of damaged composite structures will be proposed, particularly in relation to armament. The various ASTM efforts to develop fracture tests of composites will be summarized, with emphasis on finite element calculations of stress intensity factors for laminates in various test configurations.

for presentation at:

Workshop on Finite Element Methods for Composites

Imperial College London 5/6 Sept 1991

FINITE ELEMENT RESULTS FOR COMPOSITES IN ARMAMENT APPLICATIONS; CYLINDERS AND FRACTURE TOUGHNESS

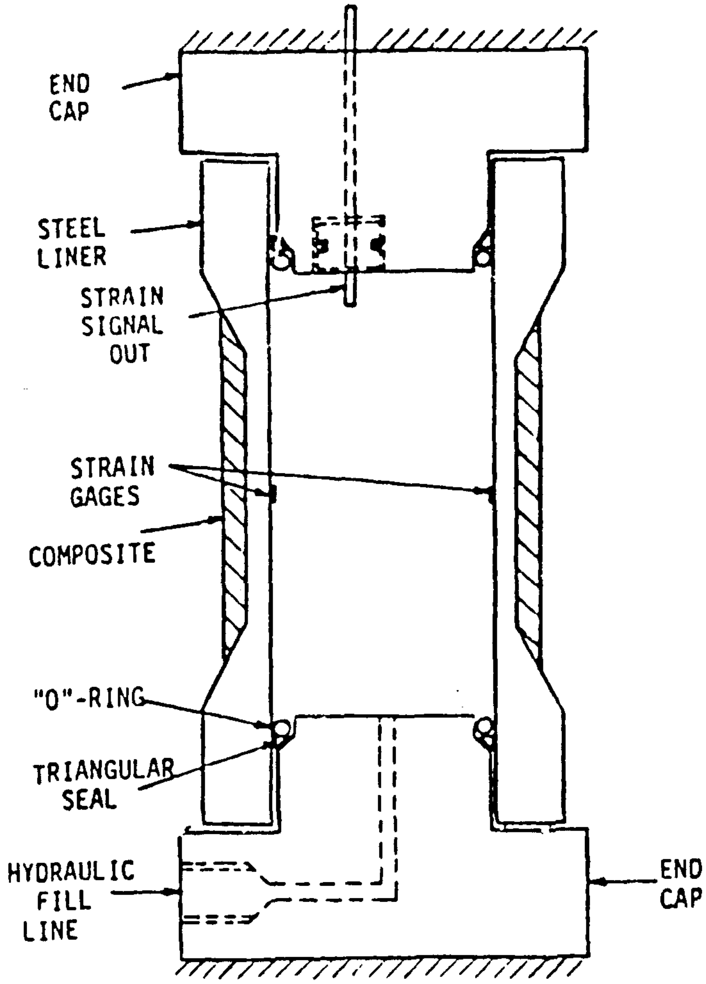
J.H. Underwood and M.D. Witherell
Army Armament RD&E Center Watervliet, NY 12189

COMPOSITE JACKETTED CYLINDERS:

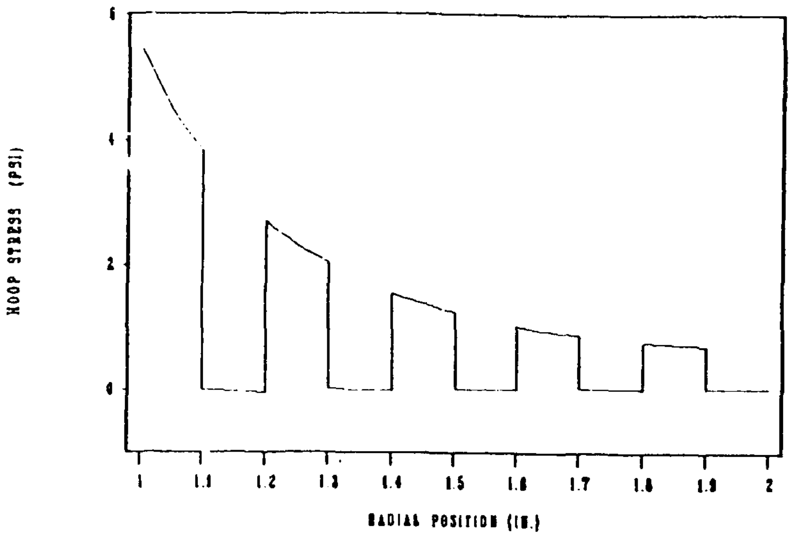
- application: light weight jacket over alloy steel
  - titanium alloy; brittle !
  - carbon/polymer; higher stiffness/weight and toughness
- multi-layer composite cylinders
  - use a modified Lekhnitskii analysis
  - close agreement with ABAQUS results
  - can model interference residual stress

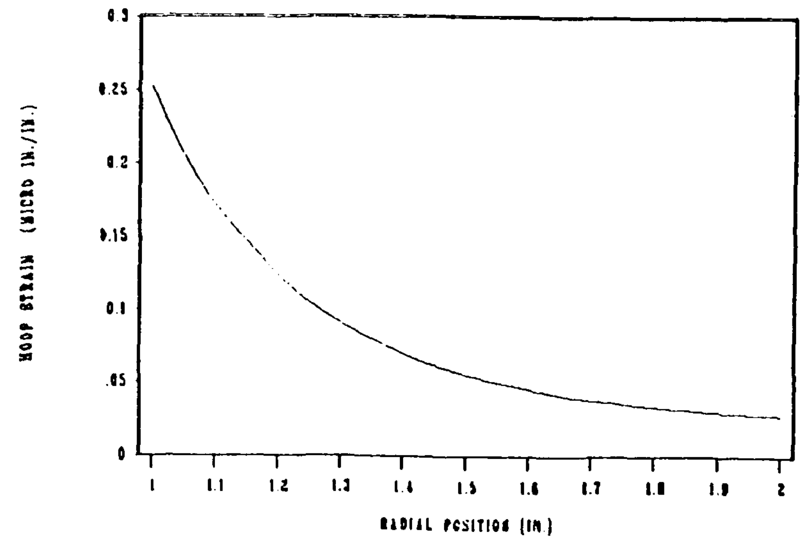
TRANSLAMINAR FRACTURE TOUGHNESS:

- application: composite structures on battlefield
  - projectile impact / penetration
  - translaminar fracture vs interlaminar delamination
- Harris & Morris approach
  - FE calculations of K-applied at notch
  - tension, compact and bend specimen configurations
  - orthotropic model of carbon/polymer laminates
  - K-ortho = K-iso ; except in tension

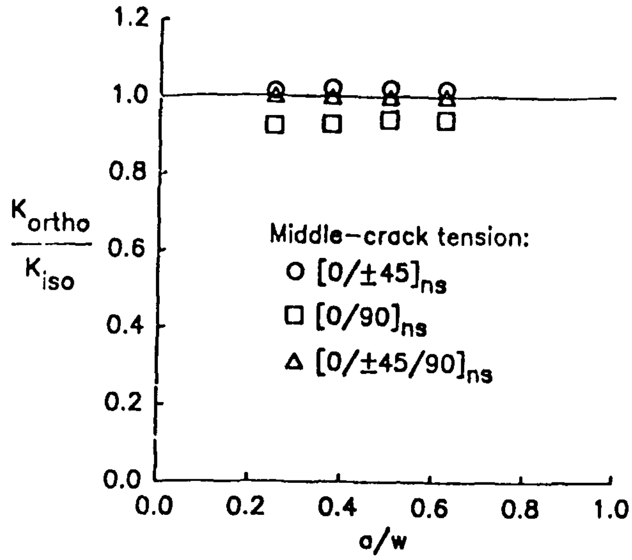


Schematic of the test setup used to measure bore strains in compound cylinders.

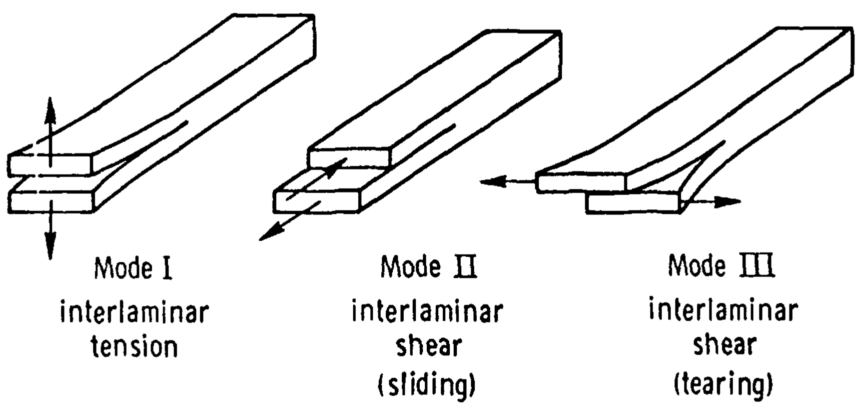

HOOP STRESS (PSI)
RADIAL POSITION (IN.)


HOOP STRAIN (MICRO IN./IN.)
RADIAL POSITION (IN.)

RATIO OF STRESS-INTENSITY FACTORS FOR ORTHOTROPIC AND ISOTROPIC MATERIAL FROM FINITE-ELEMENT ANALYSIS

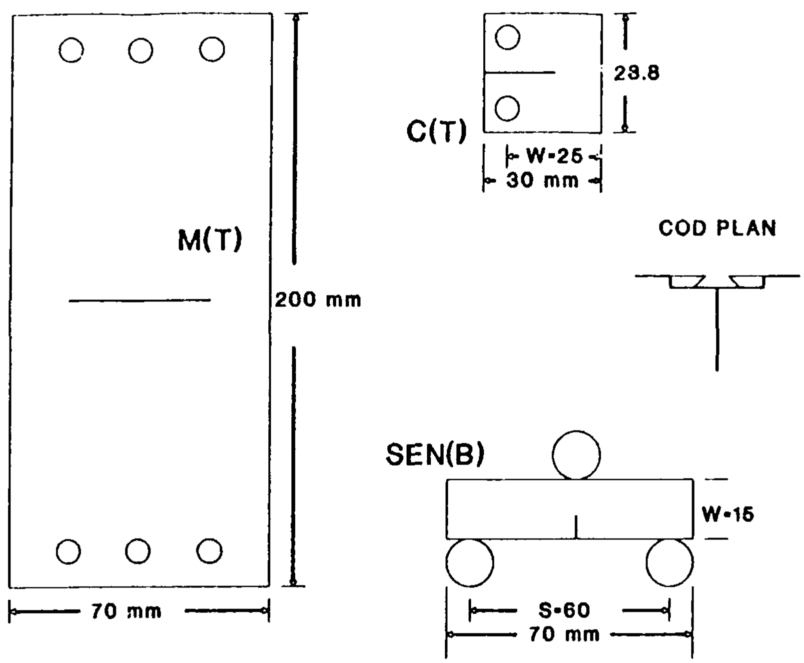


DELAMINATION FRACTURE MODES



Translaminar Fracture / E24.07.02

PLANS: Specimen Configurtations



FAILURE MECHANISMS AND MODELLING

Ozden O. Ochoa
Center for Mechanics of Composites
Texas A & M University
College Station, Texas 77843-3123

ABSTRACT

Structural response of composite components presents a major challenge in assessing and modelling damage mechanisms as a function of load. Quite often damage modes are present as a byproduct of manufacturing cycles prior to any structural loading. The task of identifying the present anomalies and numerically capturing their growth is of paramount importance and difficulty.

Following is a brief discussion on three different problems to illustrate a practical procedure in integrating present defects in numerical approximations. These cases represent modelling of a cutout section in the web of a stiffened panel, a loaded hole with drilling defects and a unit cell concept associated with textile architecture of woven prepreg.

Cutouts in a Stiffened Panel

An area of critical concern for load-carrying members is cutouts which may provide a passage for hydraulic lines or avionic harnesses, or on a larger scale, an access door in an aircraft fuselage. Parametric studies are undertaken to reflect the differences in the selection of stacking sequences, cutout size, and geometry. Primary emphasis is on the identification of localized high stress and strain in order to warn the designer of potential hazards. In this study, Patran II software is used to develop the nodal and element geometry, then converted to a neutral file. The neutral file is used as input to Abaqus, a general purpose code that is a robust performer in nonlinear behavior.

The response of a hat stiffened panel with a cutout in the web section subjected to bending loads is considered. Since tensile loads are primarily sustained by the flat panel and not transferred into the stiffener, out of plane loading is selected to observe the effect of loads on the web. Therefore, bending moments along the panel axis are imposed to induce a stress field across the cutout region. This loading is representative of the in-flight loading incurred by a stiffened wing skin. Shell elements are used in the modelling throughout the structure. The skin configuration is a $[-45/45/90/-45/0/45/0/-45/0/45/0]_S$ laminate. The stiffener web is three layers of $\pm 45°$ fabric, with a 12.7 mm diameter cutout through each web center. The cutout is positioned 79.4 mm from each panel edge, and is located 25.4 mm vertical from the panel. The flange is reinforced with 0° plies, i.e.; [45/-45/0_6/45/-45/0_6/45/-45].

The web section predominantly carries shear stresses throughout. The stress results for circular and a diamond cutout in the web supports the use of diamond shaped cutouts in regions of a structure which are primarily in shear loading, and contain primarily ±45° fibers. It can be deduced that for an effective composite design, a dominant percentage of fibers should be aligned with the principle stress direction. In addition, if a cutout is required in the design, it is most desirable to implement a rectangular cutout which contains sides parallel and perpendicular to the principle stress directions.

Hole Quality and Performance in Joints

Frequently, drilling causes delaminated regions around the circumference of the hole. The propagation of these interlaminar cracks, or delaminations, is one of the most serious problems in failure prediction of composite parts. Actual damage zones caused by drilling can be documented through the use of both X-ray and ultrasonic C-scans and optical micrographs. From the study of optical micrographs, it is observed that the lower 30% to 40% of the coupon, drill exit side, contains delaminated layers. Therefore in the numerical model, they will be assigned reduced moduli. Furthermore, as an approximation, we will treat the damage regions as circular in shape and centered about the center line of the hole with effective delamination diameters of 1.5·D and 2.0·D.

A pin loaded coupon under tension load is simulated with a finite element model. The pin is represented by distributed loads applied to its contact surface. Thus rather than model the pin itself, the effects of the pin are represented as an evenly distributed pressure. Since the coupon geometry, assumed load distribution, and assumed delamination shape are symmetrical about the long axis of the specimen, a half symmetry model is generated. Finite element analysis is performed microcomputer with a MS DOS-based program, Algor. A four node composite plate element based on the assumptions of classical plate theory is used. Each node has five degrees of freedom: two orthogonal in-plane displacements (u,v); one out-of-plane displacement (w); and rotations about two orthogonal in-plane axes ($\partial^2 w/\partial x^2$, $\partial^2 w/\partial y^2$).

Despite some short comings in the modelling assumptions, these two-dimensional models give insight into the conditions surrounding the hole. Through a layer by layer representation of the lamina stress, these models give a better understanding of the damage created in the coupons during testing. The results describe the stress distribution around the hole allowing comparisons between stacking sequence, load levels and drilling induced damage levels.

Woven Prepreg/Laminate

In brittle material systems, parameters of importance in modelling constitutive behavior are largely driven by processing methods. For example, shrinkage observed during processing results in a laminate which exhibits both matrix cracks and weak interfacial bonds. It is important to assess the status of a laminate prior to mechanical and thermal service loads. However, it may be improper to assume that the above cited damage is always detrimental. A strong emphasis need to be placed on obtaining statistical data to support the modelling of possible defects within a unit cell of the woven laminate. The goal is to illustrate the changes in the moduli (constitutive relations) as a function of applied load.

The numerical models that follow address the response of thin laminates, i.e., six layers of 8-Harness satin weave woven laminates, subjected to static loads. After careful studies of the surface with SEM, the unit cell is approximated as a simple square with 2300 micron edges. A bundle crossover square region of 800 microns in the center is modelled with a typical adjacent debond of length 240 microns. Since the material in the immediate vicinity of the crossover region consist of warp and fill yarns, it is treated as a homogeneous specially orthotropic material. The material properties of bundle crossover section is also treated as specially orthotropic , however they are modified as a function of the crimp angle.

Four node shell elements, S4R, of ABAQUS with nonlinear geometric option and a user material subroutine developed by the author is used. This specially developed subroutine enables the incremental analysis to check for failure strain in each element, at each integration point and at each load level. For the tensile loads, symmetry boundary conditions are applied to a quarter model. The tensile stress-strain results indicate that the load carrying ability is significantly diminished in the presence of a debond. Since the actual laminate consists of many layers of the woven fabric with random defects, it is appropriate to calculate an effective elastic modulus from the unit cell analysis that can be used as a layer modulus in structural response models.

USE OF ABAQUS TO MODEL COMPOSITE STRUCTURES AND MATERIALS

Michael R. Wisnom

University of Bristol, Department of Aerospace Engineering

1. Introduction

ABAQUS has been used at Bristol University for a wide range of composite problems from analysis of complete structures down to micromechanical modelling at the level of individual fibres. In this paper a number of applications are presented and the way that ABAQUS was used is discussed. Special features that make ABAQUS suitable for the analysis of composites are described, as well as some of the limitations and problems encountered.

2. Cruciform Joint

The first application is the analysis of thermoplastic matrix-carbon fibre cruciform joints. This was a conventional structural analysis which illustrates some of the features ABAQUS has for modelling composites. Layered shells were used, and these are available as 4 noded or 8 noded elements. Many finite element programs calculate equivalent orthotropic properties for membrane, bending and membrane-bending coupling, and this limits them to linear elastic analysis. However ABAQUS uses numerical integration through the thickness, which allows non-linear material response to be modelled in the layers. These elements also allow transverse shear flexibility.

Linear elasticity was assumed in this particular analysis. A static analysis was carried out of the joint under tension loading. Thermal residual stresses were included by imposing a temperature change. The analysis was relatively straight forward, and no particular problems were encountered. A large amount of output data was produced of layer stresses and strains. These can be output at integration points or averaged at nodes. One disadvantage is that interlaminar shear stresses are not computed. However shear forces can be recovered in the output file and interlaminar stresses calculated in a post-processing routine. Another limitation is that standard failure criteria are not available, although again these can be included in the post processing.

3. Pin Loaded Plates

Another application is in the analysis of failure of pin-loaded silicon carbide-aluminium laminated plates loaded in tension [1]. The local stress distribution is determined by the contact between the pin and the hole. ABAQUS has conventional gap elements to model contact, however it also has interface elements which are particularly useful. The advantage of these elements is that it is not necessary to use local coordinate systems to define the direction of the interface. Also interface stresses are calculated rather than forces.

The stress-strain response of the material is non-linear, especially in shear. This was modelled using the user defined subroutine UMAT. For a given strain vector it is necessary to define the tangent stress-strain matrix and the stress vector. The stress-strain response of the unidirectional material in the transverse and fibre direction in both tension and compression and also the shear response were measured and the data was curve fitted. The orthotropic stress-strain matrix was then calculated by interpolating the data. The response was assumed to be elastic. In practice there is plastic deformation, but for monotonically increasing load this will not affect the results. This approach assumes that the response in the transverse direction, fibre direction and in shear are independent. For example the non-linearity in transverse compression is assumed not to be affected by the shear stress. In practice there is some interaction, but this is a simple approach which allows a reasonable approximation of the complex behaviour of the material.

Within the UMAT routine it was also possible to include a number of different failure criteria. Partial failure could also be modelled. For example when the transverse tensile stress in one layer reached a critical value the transverse and shear moduli could be set to zero, whilst still retaining the fibre direction modulus. This allowed failure to be tracked from initiation right through until catastrophic fracture. Comparison with test results indicated a reasonable correlation.

4. Four Point Bending Tests

Four point bending of unidirectional carbon fibre-epoxy was analysed in order to quantify the errors involved in applying linear elastic bending theory to deduce the stress at failure from the applied load [2]. A two dimensional analysis of a slice through the specimen was carried out. Very large deformations occur in these tests and so the non-linear geometry option was used. Due to the large rotations, the angle of the forces at the rollers changes significantly. Also the specimen slips on the rollers and so the point of contact moves along the specimen. Both of these effects can be modelled using the rigid surface capability in ABAQUS. The rollers are defined as rigid surfaces and interface elements are used to connect them to points on the specimen where contact may occur. At each iteration all these interfaces are checked for contact and so the actual points of contact do not have to be defined beforehand. This worked very well, although it was found that a very fine mesh was needed in order to avoid step changes in results as contact shifted from node to node along the specimen.

Friction was also included at the rollers. This worked well for small values of friction coefficient, however for larger values convergence became slow. The friction algorithm makes use of a stiffness in stick, the value of which is hard to determine. If too low a value is used significant relative movement can occur whilst the interface is supposed to be sticking. Alternatively, if too high a value is used convergence can become difficult. This problem should be solved shortly with the introduction of a new friction algorithm.

Non-linear material properties were required, in this case to take account of the changing modulus with strain in the fibre direction. This was modelled with an orthotropic non-linear elastic UMAT routine, similarly to the metal matrix plate analysis. Because of the large displacements, rotation of the material axis system with the specimens is essential. ABAQUS handles this provided the *ORIENTATION option is used. Stresses are also output in this rotated system. Strains of over 2% arise in tests on carbon fibre-epoxy and with glass fibre-epoxy strains of over 5% can occur. Whilst ABAQUS can handle such large strains, it should be noted that errors can arise in using a standard elasticity formulation at large strains. These errors are typically of the same order of magnitude as the strain. Continuum elements in ABAQUS use true strain rather than engineering strain and at large strains the difference between these strain measures also starts to become significant.

This analysis generally worked very well. However one problem arose when a similar analysis was performed with shell elements. It was found that under pure bending small fictitious axial forces were generated. The same phenomenon occurred with beam elements. Although the magnitude of the forces was small, they were significant since the whole purpose of the analysis was to investigate deviations from standard bending theory. This illustrates the care that has to be taken in any analysis to adequately check and understand the procedures being used by using simple test cases.

5. Delamination of Tapered Specimens

Delamination in tapered unidirectional glass fibre-epoxy with dropped plies has been analysed [3]. A plane stress analysis of a slice through the specimen was carried out using orthotropic elastic properties. In the taper, the fibres curve around the end of the terminating plies and so there is a continuous change of fibre direction. This effect can be modelled very easily using the user subroutine ORIENT to define the material axes, but would cause difficulties without this option.

The stresses in the specimen subject to tension were calculated and then the strain energy release rate was evaluated assuming delamination above and below the terminating plies propagating into the thick section. The energy release rate was calculated from the difference in the total strain energy in analyses before and after propagation of the delamination.

It has been found that the resin layer between plies affects the delamination significantly. This effect has been modelled using non-linear springs to represent the elastic-plastic behaviour of a thin resin layer. Satisfactory results were obtained, however when energy release rate calculations were performed it was found that the strain energy was incorrectly calculated for the springs. It was therefore necessary to use the forces and displacements to calculate the energy, and this was incorporated in a post processing routine.

6. Compressive Failure Mechanisms

The compressive strength of unidirectional composites depends on the shear stiffness of the composite which prevents buckling of the fibres. Another key parameter is fibre misalignment. This causes shear stresses and hence shear strains which result in an increase in misalignment. At a certain critical compressive stress this process becomes unstable and failure occurs. ABAQUS was used to simulate this failure mechanism [4]. Initially it was thought that orthotropic material properties could be used with the non-linear geometry option. However this was found not to work because the material axes only rotate with rigid body rotation, but not with shear strain. An alternative approach was used based on REBARs to model the fibre stiffness. These are forced to maintain their position relative to the element and so the fibre direction effectively changes with both rigid body rotation and shear strain, as desired.

The material properties of the continuum elements containing the REBARs were chosen to represent the transverse and shear properties of the composite. The non-linear shear response is critical, and so this was modelled using a UMAT subroutine, as before.

This modelling approach worked satisfactorily, and ABAQUS was able to simulate the instability leading to compressive failure. The effect of misalignment angle on compressive strength was investigated and shown to be crucial when the composite is subject to a uniform stress field with no constraint. However when the constraint imposed by the test fixture is included, the effect of misalignment is greatly reduced.

7. Micromechanical Modelling of Metal Matrix Composites

A slice through a section of unidirectional silicon carbide-aluminium was modelled using generalized plane strain elements [5]. These allow a constant but non-zero strain in the fibre direction. A quarter fibre with surrounding matrix was modelled, with boundary conditions to represent an infinitely repeating array of fibres. Elastic-plastic matrix properties were used, with a Von Mises yield criterion. Initially isotropic hardening was used, but later the kinematic hardening option was adopted as this is a better model when reversed loading is involved. The fibres were assumed to be elastic. Initially the cooldown from manufacturing was analysed. This requires temperature dependent material properties to be defined, which can be done in ABAQUS. Later it was found that this was unnecessary, and all that was needed was to apply the total differential thermal strain and use the room temperature properties. Before analysing the composite under load, it was necessary to alter the matrix material properties to account for the age hardening that takes place after manufacture. This could be done in ABAQUS during the analysis by using the *FIELD option or by defining a fictitious temperature dependence of properties.

A user defined element was written using the UEL subroutine in order to model the interface. This allowed interface failure to be modelled using an interaction equation between normal and tangential stresses. It also allowed a special solution procedure to be implemented to overcome convergence problems during unstable interface debonding [6].

Transverse tensile loading of the composite was then analysed and factors affecting the strength investigated. It was concluded that the most critical parameters are the interface strength and the level of residual stresses. The latter are beneficial to transverse strength because they are compressive across the interface, effectively increasing the interface strength.

8. Concluding Remarks

ABAQUS has been used in a wide range of applications and has been found to be very good for modelling composites. It is especially suitable for non-linear analysis. A good range of standard capabilities is available, and these can be extended by means of user written subroutines when necessary. This offers the best of both worlds in enabling the user to benefit from a powerful, general purpose, commercially supported program whilst being able to implement specific capabilities where required via user subroutines. Whilst ABAQUS has limitations, and some problems have been encountered, it is a program undergoing rapid development, and the suppliers, Hibbitt, Karlsson and Sorensen Inc., have generally been found to be responsive to problems that have arisen.

References

1. Graham, U., "Experimental and theoretical analyses of un-notched and pin loaded metal matrix composite laminated plates", Ph.D. thesis, University of Bristol, Department of Aerospace Engineering, to be submitted.

2. Wisnom, M.R., "Limitations of linear elastic bending theory applied to four point bending of unidirectional carbon fibre-epoxy", Proc. AIAA Structures, Structural Dynamics and Materials Conf., Long Beach, April, 1990, pp 740-747.

3. Wisnom M.R. "Non-linear analysis of misaligned unidirectional carbon fibre-epoxy compression specimens", to be published.

4. Wisnom, M.R. "Delamination in tapered unidirectional glass fibre-epoxy under static tension loading", Proc. AIAA Structures, Structural Dynamics and Materials Conf., Baltimore, April, 1991, pp 1162-1172.

5. Wisnom M.R. "Factors affecting the transverse tensile strength of unidirectional continuous silicon carbide fibre reinforced 6061 aluminium", J. Composite Materials, Vol. 24, July 1990, pp 707-726

6. Wisnom M.R., "Modelling fibre-matrix interface failure in unidirectional metal matrix composites", presented at ABAQUS Users Conference, Harwell, September, 1989.

CONSIDERATIONS ON THE USE OF FINITE ELEMENTS IN THE ANALYSIS OF COMPOSITE STRUCTURES

By W.S.Arnold, I.H.Marshall, J.M.Thompson and J.Wood
Department of Mechanical and Manufacturing Engineering
Paisley College

1. INTRODUCTION

Due to certain intrinsic factors associated with composite materials (for example their laminar orthotropic nature), many problems are extremely difficult to study using classical techniques or are completely intractable with no closed form solutions available. Using Finite Elements to analyse such problems can often appear an attractive alternative, however, there are several distinctions which have to be acknowledged when using the technique in comparison to the analysis of homogenous isotropic materials.

This paper outlines some of these considerations with reference to element types and modelling strategies available in commercial F.E. packages. Some initial thoughts on benchmarks for composite elements are also presented.

2. ELEMENT TYPES

As with conventional materials the correct choice of element type is critical. Clearly, the element formulation should be appropriate to the type of problem and the analyst should be aware of its limitations in terms of displacements and in particular stress output. Four broad categories are available in commercial systems.

(1) Laminated plate/shell elements ; These elements are used for the Representative Volume Element (RVE) modelling of laminated plates and shells.

(2) Sandwich plate/shell elements ; These elements are used to model 'sandwich' type structures such as foam filled plastics and honeycomb metallic structures.

(3) Orthotropic brick ; This element could be used for RVE modelling of a laminate or ply by ply modelling in a Replica analysis. However, the element would not allow individual ply stresses to be obtained in an RVE model, consequently its main use is in Replica modelling, however the element is widely used for structural stiffness assessment and dynamic analysis.

(4) Laminated or stacked brick ; This element is used for the RVE modelling of laminated composites. Interlaminar shear and normal stresses are normally available as output. However the accuracy of these quantities shall be a function of the number of plies per element.

3. MODELLING TECHNIQUES

Various levels of analysis are possible, from the microscopic level in which the composite constituents are analysed to a ply by ply analysis in which the plies are modelled as distinct entities through to a macro analysis where equivalent properties are evaluated for the laminate, it is then treated as an orthotropic homogeneous material.

The type of model required for a particular analysis is very much dependant on the engineering requirements. Various modelling techniques utilised in the analysis of composite structures are shown in Figure 1.

(1) Representative Volume Element (RVE) Models ; This approach is based on the assumption that the stress response of a composite can be determined by the volume average response of an element. The properties of the RVE element at the macro level are derived from individual fibre/matrix/ply properties, using lamination theory, the rule of mixtures or some other micro mechanics theory. Alternatively, these properties can be determined from various tests performed on representative specimens of the material.

The RVE approach generally simplifies the analysis problem and allows ply stresses to be obtained as a post processing operation from laminate strains. However this approach is not generally employed to analyse detailed conditions such as 'free edge effects', damage modelling or geometrical discontinuities such as ply drop offs.

A number of 'hybrid' RVE elements have been derived which allows the stress singularity at free edges to be modelled. However none of these appear to have been implemented in commercial finite element systems.

(2) Replica Modelling (3-D Modelling) ; In this approach, laminate plies or in some cases discrete fibre/matrix models are represented by a series of unlaminated orthotropic elements, perhaps as many as two or three parabolic elements per ply. Obviously, this level of analysis cannot be used for large structural problems due to computational limitations but is of value when looking at detailed interlaminar stress distributions and damage mechanics.

(3) Hybrid Modelling ; With this approach, volume average property distribution modelling (RVE) is combined with discrete ply modelling (Replica) in the same analysis to create global/local models. Care must be taken in the selection of the global/local interface assumptions. With this approach, stacked bricks or laminated plate plate/shell elements can be used in conjunction with a local ply by ply model.

(4) Nested Modelling ; This is exactly the same approach adopted traditionally for the analysis of detail in isotropic structures. An area of a coarsely meshed model is isolated and re-meshed with either a finer mesh of RVE elements or a new mesh of Replica elements. The boundary conditions applied to the nested model are obtained from the results corresponding to the boundary region in the coarse model.

The most managable level of analysis involves the use of plate/shell elements to model the 'two dimensional' response of structures. This type of analysis can be used to predict the overall structural stiffness and in-plane stress distribution but in most cases not its failure. This is because failures tend to be initiated at discontinuities which require a lower level of analysis. However, it is worth mentioning that some semi - empirical methods have been developed to predict failure at SCFs which use 'two-dimensional ' elements, e.g. the Characteristic Distance approach by Whitney and Nuismer [1].

4. OTHER CONSIDERATIONS

Classical lamination theory, which is based on a plane stress assumption, cannot represent the interlaminar shear and normal stress components. Although these stresses may be small in relation to the in-plane stresses, the strengths associated with them are also invariably much smaller. These components of stress are therefore extremely important in causing the failures which are unique to composite materials.

Normally, the only way of accurately obtaining these quantities is to use a ply by ply Replica model or a hybrid or nested approach. However most commercial finite element systems will attempt to approximate the interlaminar shear stress by manipulating the resulting displacements and strains.

For homogeneous isotropic materials the application limits of Kirchhoff plate theory (transverse shear effects ignored) are relatively well known and applicable to thin plates. For thicker plates the inclusion of transverse shear effects is included, e.g. Mindlin type elements which assume constant transverse shear through the thickness. Very thick plates must be modeled using three dimensional brick elements.

Composite materials however, exhibit E/G ratios which may be an order of magnitude greater than in the isotropic case. With a relatively low shear modulus, it is known that the level of transverse shear deformation is important in determining maximum deflections, vibration natural frequencies, buckling loads and stress results [2]. In most cases, plates which would normally be categorized as being 'thin' require transverse shearing effects to be included for accurate results.

5. BENCHMARKS

With the many claims made by software vendors concerning element performance it is important, as a matter of quality assurance that valid benchmarks for composites are established. Agencies such as NAFEMS are currently addressing this issue [3]. There are several desirable features which benchmarks should possess, viz – (a) as simple as possible (b) only one variable at a time should be considered (c) consider only clearly valid targets either from theory or from converged Replica modelling.

Furthermore there are several generic features which benchmarks should address, including ;

– effect of lack of inter–element fibre continuity as a results of the element material model or as a result of geometrical approximation.

– membrane/bending/twisting coupling for unsymmetrical layups

– ability to model SCF's in orthotropic materials due to the high stress gradient present in orthotropic structures

– in general, composite benchmarks should conform to isotropic benchmarks.

– the recovery of interlaminar shear and normal stresses particularly for RVE models.

– examine the effect of material aspect ratios

Clearly, other important areas should be addressed, including dynamic, thermal and non –linear material models. Two simple examples are now considered which highlight some of the above features.

EXAMPLE A MEMBRANE RESPONSE OF COMPOSITE CYLINDERS

As shown in figure 2, the problem consists of an open ended cylinder under internal pressure. Two models are considered. In the first, a quarter model is used to represent an orthotropic cylinder whose principal material direction is in the hoop direction and is coincident with the element axis. In the second case, a wound cylinder in which the principal material direction is rotated from the element axis is represented by a full model. The analysis was carried out using P/FEA [4]. Two element types were considered, the QUAD/8 doubly curved shell with transverse shear effects and the linear QUAD/4 shell.

In both models only one geometrical patch was used every 90 degree segment to generate the elements thereby avoiding any disparity due to geometrical inaccuracies. Also in all cases the element aspect ratios were held constant.

Case 1

The object of the first model is to establish the sensitivity of the hoop stresses to the number of elements subtending the 90 degree section for both an isotropic and orthotropic material. Results are shown in Table 1.

Only two QUAD/8 elements are required to reach the target value while the QUAD/4 results converge towards the target with eight elements. Both material models give identical results with each element type indicating that there is no additional effect of inter-element fibre continuity in the case where the principal material direction is coincident with the element axis. The poorer results of the linear elements were expected due to the segmented representation of the cylindrical surface. This is designated method 1.

Case 2

The object of this model is to determine the sensitivity of the material stresses to the number of elements subtending 90 degrees for the case where the principal material direction is rotated with respect to the element axis. This is designated method 2.

Two methods are available in the Patran to model material directions. In the first, the material can be defined as a one ply laminate in which the principal laminate direction is coincident with the element axis i.e. the hoop direction. The laminate is then defined to be composed of one ply orientated at an angle of 60 degrees to the element axis.

In the second method the principal laminate direction is defined as lying at 60 degrees to the element axis . The laminate is then defined as being composed of one ply orientated at 0 degrees to the laminate direction. Results are shown in table 2.

Using the QUAD/8 element and method 1 to generate the material directions, gives the target values for model WCYL2LO. Employing method 2 however, gives errors of 14% and 5% on the material stresses. Results improve significantly by doubling the number of elements around the circumference. Results for the QUAD/4 elements again show disparities between the two methods. Doubling the number of elements gives only minor improvements in stress with respect to the target values.

These apparent dispcrepancies are currently being investigated by the software suppliers. This example serves to illustrate that simple checks of this nature can indicate basic innacuracies in the modelling of material properties. Bearing in mind the low transverse strengths of composites, even small errors in stress can be important. Additionally, in this example the cylinder is essentially subjected to a uniaxial stress field. The previous disparities may be more acute in shells subjected to bending or at SCF's.

EXAMPLE B BENDING OF A TWO LAYERED COMPOSITE BEAM

This example consists of a two layered cantilever beam subjected to an end load and applied couple as shown in Figures 3 and 4. The problem has been solved by Lekhinitskii [5] who provides bending stress and shear stress distributions through the beam thickness thereby enabling direct comparisons with F.E. solutions.

As the theoretical model is based on Kirkhoff theory, the restraints do not fully fix the end nodes but allow for transverse deformation relative to the neutral axis. The ratio of Young's Moduli in the longitudinal direction of the beam for the two layers is 9E1 = E2. The analysis was carried out using PAFEC.

Results are plotted in Figure 5 for the bending stress and interlaminar shear stress. As can be seen the results are in excellent agreement. The high mesh refinement used in this model is required for accurate retrieval of the interlaminar shear stresses. Accurate bending stresses were recovered using models with only three elements through the thickness.

It is considered that this example would make an excellent benchmark for Replica modelling problems where the accuracy of bending and interlaminar shear stresses are important parameters in the analysis.

6. CONCLUDING COMMENTS

Various modelling strategies and observations on the importance of shear deformation in composite behaviour are indicated. Important aspects on benchmarks for composite elements are discussed and two sample problems are presented.

REFERENCES

[1] Whitney, J.M and Nusimer, R.J. J.Comp.Mater 8, 1974, p 253.
[2] Vinson,J.R. and Sierakowski,R.L 'The Behaviour of Structures Composed of Composite Materials', Marti Martinus Nijhoff Publishers, 1987.
[3] Taig,I.C. Finite Element Analysis of Composite Materials, Ian Taig Engineering Technology, Report Number R 90/3. DTI Contract NEL/F/675.
[4] P/FEA User Manual, Release 2.4, 1989, PDA Engineering 2975 Redhill Avenue, Costa Mesa, California .
[5] Lekhinitskii,S.G. Anisotropic Plates, Gordon and Breach Science Publishers, 1968.
[6] PAFEC Data Preparation Manual, Pafec Ltd, Nottingham, 1988.

MODEL	ELEMENT	EL/90 degree	MATERIAL	TARGET	RESULTS
CYL2I	QUAD/8	2	ISO	10	10
CYL2AI	QUAD/4	2	ISO	10	9.24
CYL4AI	QUAD/4	4	ISO	10	9.81
CYL6AI	QUAD/4	6	ISO	10	9.92
CYL8AI	QUAD/4	8	ISO	10	9.96
CYL20	QUAD/8	2	ORTH	10	10
CYL2AO	QUAD/4	2	ORTH	10	9.24
CYL8AO	QUAD/4	8	ORTH	10	9.96

QUARTER MODEL RESULTS – EXAMPLE A TABLE 1

MODEL	ELEMENT	EL/90 degree	METHOD	TARGET		RESULTS	
				L	T	L	T
WCYL2LO	QUAD/8	2	1	2.5	7.5	2.5	7.5
WCYL20	QUAD/8	2	2	2.5	7.5	2.14	7.86
WCYL40	QUAD/8	4	2	2.5	7.5	2.41	7.59
WCYL4ALO	QUAD/4	4	1	2.5	7.5	2.38	7.4
WCYL4AO	QUAD/4	4	2	2.5	7.5	2.48	7.32
WCYL8AO	QUAD/4	8	2	2.5	7.5	2.41	7.42

FULL MODEL RESULTS– EXAMPLE A TABLE 2

FINITE ELEMENT MODELLING OF COMPOSITE MATERIALS AND STRUCTURES

REPRESENTATIVE VOLUME ELEMENTS (RVE's)

- rule of mixtures
- lamination theory
- shell elements
- laminated bricks
- allows ply stresses to be obtained as a post -processing operation
- validity ?
- accuracy ?

REPLICA MODELLING

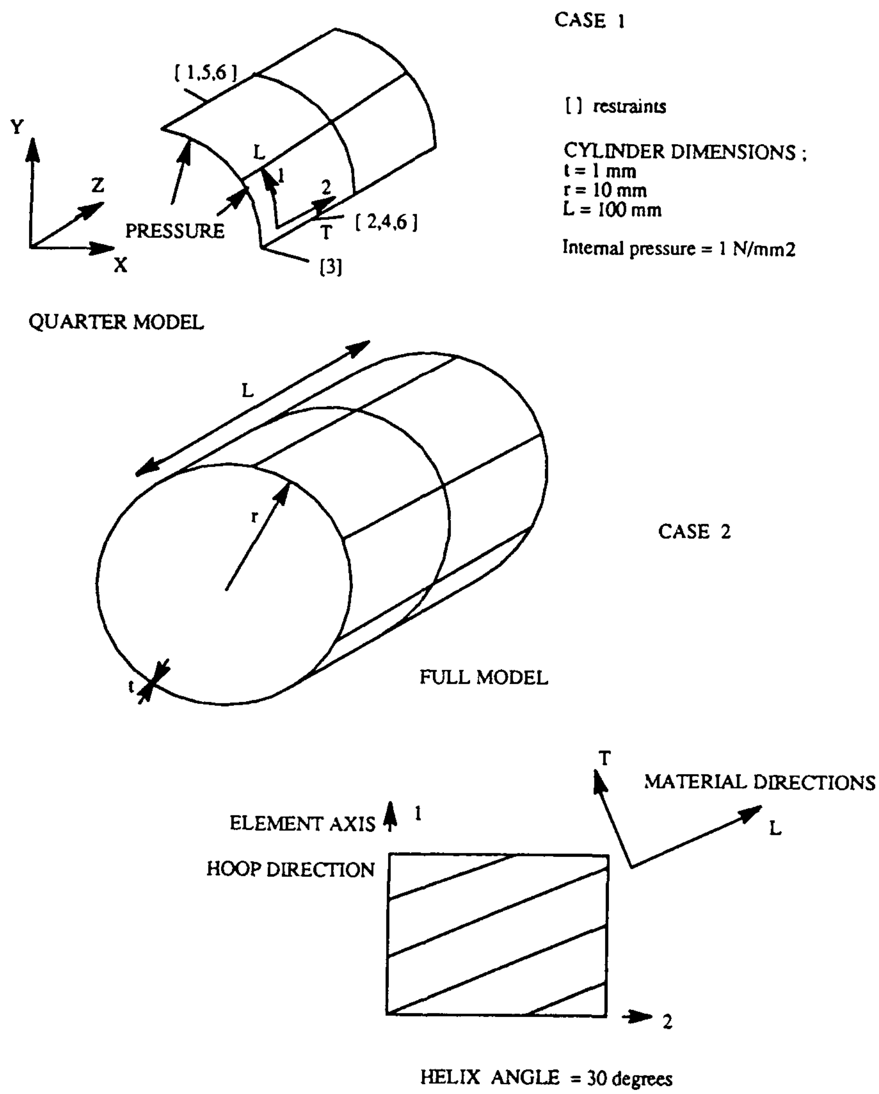
- orthotropic bricks
- ply by ply models
- 3–4 elements per ply
- post processing cumbersome
- extremely large models
- local and global effects theoetically possible

HYBRID MODELLING or NESTED MODELLING

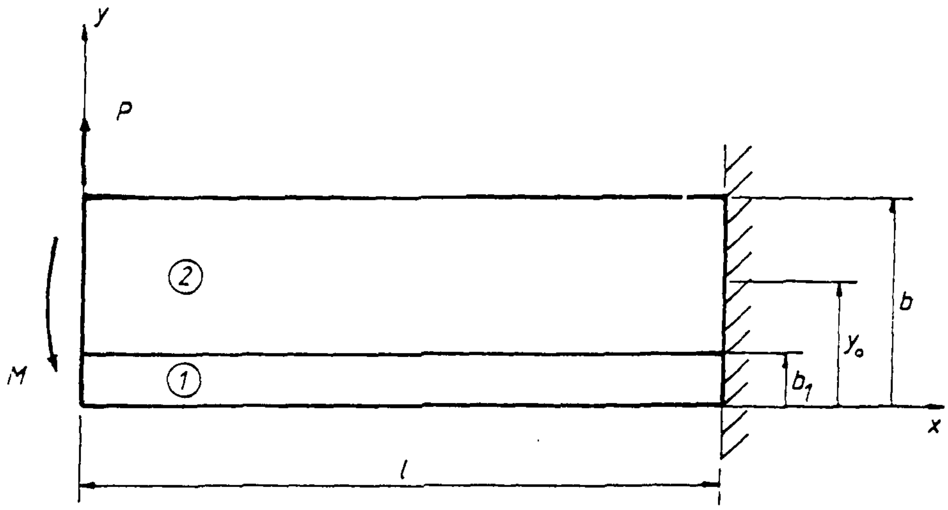
RVE and REPLICA in same model
- combinations of laminated shell, bricks and orthotropib bricks

RVE followed by local REPLICA modelling

FIGURE 1



EXAMPLE A MEMBRANE RESPONSE OF COMPOSITE CYLINDERS FIGURE 2



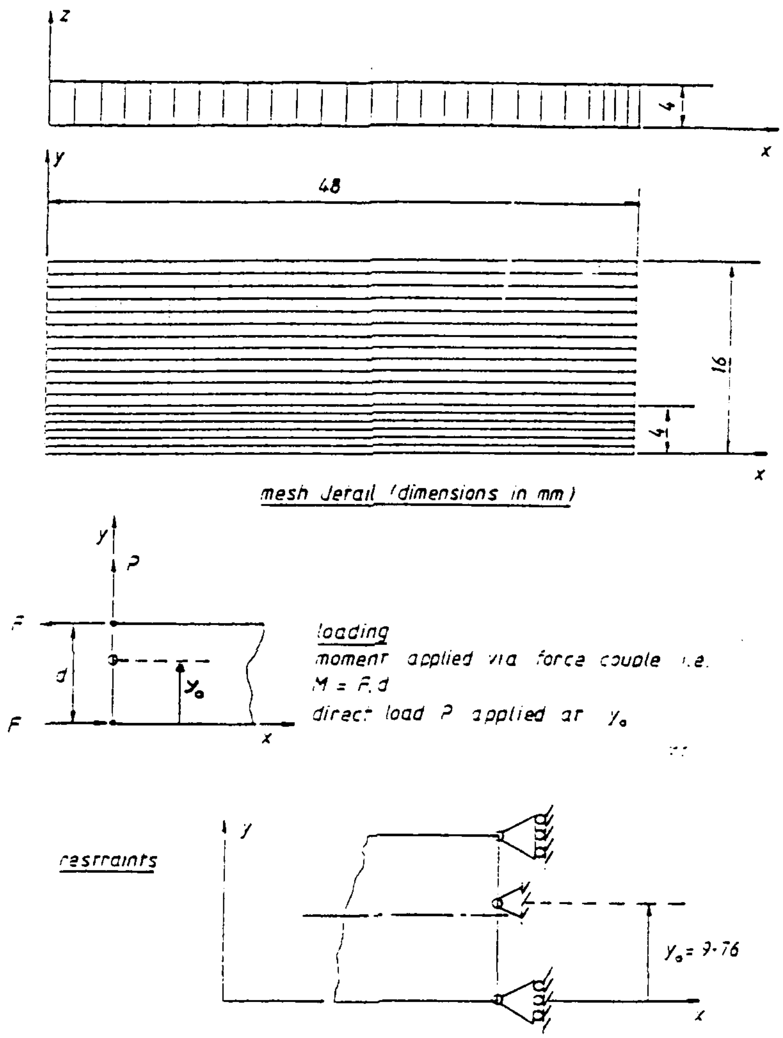
Problem specification :

$b = 4b_1$, $y_o = 0 \cdot 61\, b$ (neutral axis) , $9E_L^1 = E_L^2$

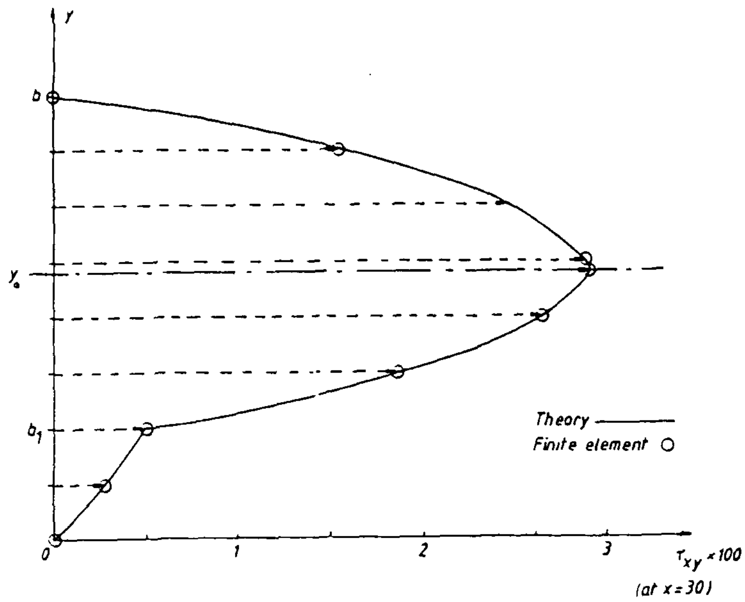
$l = 4b$, $M = P = 1$

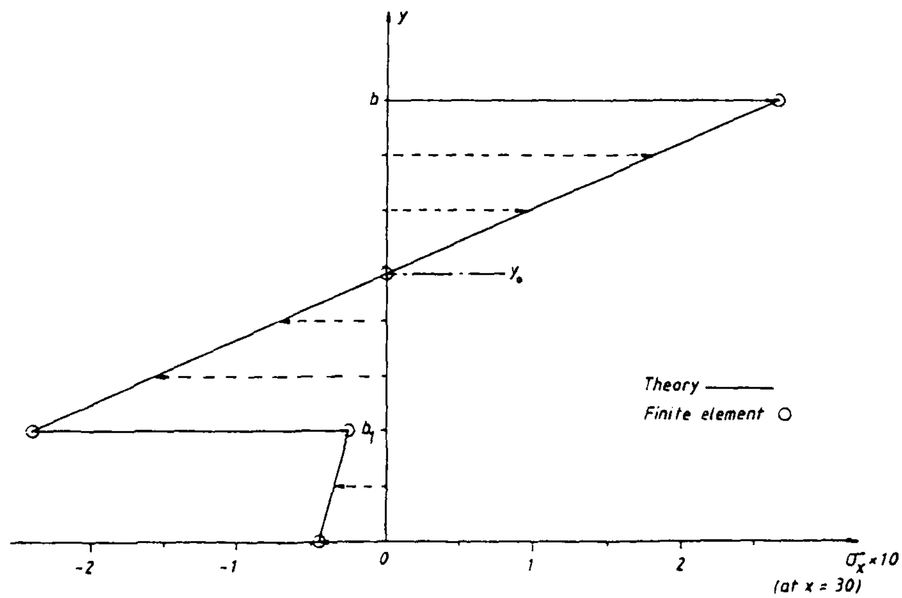
EXAMPLE B BENDING OF A TWO LAYERED COMPOSITE BEAM

FIGURE 3



EXAMPLE B MODEL DETAILS FIGURE 4





EXAMPLE B BENDING STRESS AND SHEAR STRESS DISTRIBUTION

FIGURE 5

Finite Element Analysis of Composite Space Systems

by

M J Gunn and L Hollaway

Composite structures for space

For a number of years the Composite Structures Research Unit in the Department of Civil Engineering at the University of Surrey has been concerned with the design of large structures for use in space. This type of structure may be used for space communications, space science or the establishment of the first permanent space station. Composite materials are particularly appropriate for this type of application because of their favourable strength and stiffness to mass ratios. A typical structure is a three dimensional double layer truss designed to support a parabolic reflector for communications. These structures present a number of design problems where finite element analysis may be of assistance, including vibrational characteristics, deployment from their initial (folded) form, and the effectiveness of the jointing system that is adopted.

Using more than one program

The analyses reported below have taken place over the last six years. Initially we only had access to LUSAS at the University of Surrey. At the time of writing we have LUSAS, ABAQUS, NASTRAN and ANSYS available for use. There are a number of good reasons for using more than one finite element program in research projects. Firstly programs vary in their capabilities and ease of use. Secondly, there is the question of reliability of results. Repeating an analysis using a different program checks data perpetration and understanding of how to operate program options as well as the correctness of the program itself.

The first two analyses would seem to demonstrate a general superiority of ABAQUS over LUSAS. We should empasise that we have faith in the LUSAS program and continue to use it in our research. A significant advantage of LUSAS is that it is a British program and excellent local support is available. ABAQUS is an American program and although it is probably the leading non linear code available one can expect difficulties in obtaining the same level of support as for a program originating in the UK.

Dynamic analysis of unsupported structures

It is well known that a structure must have a minimum number of supports if one is to perform a proper static analysis. A structure in space is unsupported and therefore a step by step dynamic analysis (based on Newton's laws of motion rather than static equilibrium) is appropriate. The absence of the section of data that describes the support conditions led to a data error from LUSAS. The insertion of "free" boundary conditions led to a division by zero error. The question then arose: was this a problem with LUSAS or Newmark's method? The various step by step dynamic algorithms described by Bathe (1977) were programmed and tested against a simple problem with an analytical solution. None had a problem with the absence of displacement controlled boundary conditions. In the meantime ABAQUS was discovered mounted at the University of Manchester Regional Computer Centre. One of the problems in the Examples Manual was the step by step dynamic analysis of an unrestrained structure. ABAQUS was then successfully applied to our problem (Hollaway et al, 1987). A particularly useful feature of ABAQUS was the possibility of automatically determining the size of time step required as the analysis progressed. This is done by estimating the equilibrium error at the mid point of the time step and repeating the step (with a shortened time) if this error exceeds some specified value.

Stress analysis of a joint

The second problem to be discussed is the non linear analysis of a tubular lap joint. Various analyses were carried out with tubes made from either aluminium or carbon fibre/polyethersulphone (PES) composite (Romhi et al, 1986, Romhi, 1990). One variation on the basic configuration was that the outer tube was replaced by an end cap which would enable the member to be joined to others in a three dimensional truss structure. The objective of the analyses was to examine the stress distribution in the joint and assess the effect of the different material properties on the behaviour of the joint as failure was approached. Axial loads were applied to the joint and the condition analysed was one of axisymmetric plane strain. The tubes were assigned elastic properties. The adhesive was modelled as an elastoplasitic material, yielding according to Von Mises yield condition.

Both LUSAS and ABAQUS showed yielding starting at the ends of the adhesive. Under increased load these zones of yielding spread towards the centre of the joint. At the time these analyses were performed ABAQUS had an automatic load incrementation facility similar to that described above for dynamic analysis whereas LUSAS required that the size of each load step be specified before the analysis started. As a result we had difficulties in obtaining a converged analysis using LUSAS despite several repetions with modified sizes of load increments.

Using the "User Material" facility in ABAQUS we were able to modify the yield condition for the adhesive to one where yielding depended on the mean normal pressure. This involved producing a Fortran subroutine which would calculate the elastoplastic modulus matrix relating an increment of strain to an increment of stress and in addition would ensure that stresses did not exceed yield by performing a plastic stress correction. We should issue a warning that using this type of facility requires expert knowledge of finite element programming. The work was checked by programming ABAQUS with Von Mises yield condition via the User Material facility and checking against the standard Von Mises code in ABAQUS. All the routines that we wrote were also independently checked in another (less sophisticated) finite element program.

Specification of anisotropic properties

Another research project examined the behaviour of a single composite tube under impact and general vibrational loading. Here the analysis was to be three dimensional and the first question considered was whether the tube should be modelled by this or thick shell elements. The research worker had taken care to establish the anisotropic elastic properties of the carbon and glass reinforced PES composites, but there were unexpected difficulties in using these properties in the finite element models. The most straight forward case was where the semiloof curved thin shell elements in LUSAS were used. Properties for these elements are always specified in terms of local co-ordinates associated with each element. The thick shell elements in LUSAS are basically "cut-down" versions of the familiar 20 noded isoparametric brick elements with the edge nodes removed through the thickness of shell. By default properties are assumed to be in terms of the global axis system. To enter properties aligned with the element geometry it is necessary to specify a local cartesian coordinate system for each element. This is unsatisfactory for two reasons. Firstly an extra approximation is introduced into the analysis as the curvature of the element is ignored. Secondly it is strictly speaking, unnecessary since this information has already been entered in the element connectivity information.

The use of ABAQUS was considered, but not pursued, as properties had to be entered directly as elements of the tensor D_{ijkl}. Finally a reduced from of anisotropy was used with LUSAS in conjunction with properties specified in the directions of the global axes. One global axis was aligned with the axis of the tube (the 90% fibre direction) and all properties in perpendicular planes were assumed to be equal. In practice, the results of the analyses conducted were mainly controlled by the longitudinal modulus and satisfactory modelling was achieved with isotropic properties. Hence in this case the agonizing over correctly entering anisotropic properties was unnecessary.

<u>Analysis of a plate loaded by a single pin</u>

This is the simplest form of analysis relevant to bolted joint behaviour. This problem was used to compare ease of data generation, modelling options and accuracy of calculated stresses for LUSAS, ABAQUS and MSC/NASTRAN. All three packages have internal geometric preprocessors and in summary: LUSAS's preprocessor is simple to use but less general and NASTRAN's preprocessor has the most generality but suffers from complexity.

ABAQUS and NASTRAN both have options for calculating the effective material properties according to laminated plate theory and can also output stresses in the different layers. LUSAS and ABAQUS have similar options for printing stresses at nodal points: either averaged or based on individual element values. MSC/NASTRAN, on the other hand, calculates stresses at the element corners and centre expressed in the element's local co-ordinate system. The original intention had been to use the 8-noded isoparametric element with reduced integration in each package. Unfortunately, NASTRAN does not allow a global definition of the material axes with this element type and so a finer mesh of 4-noded elements was used instead. Analyses were also conducted with 9-noded elements and the 8-noded elements with full integration.

The results were assessed in terms of the overall stiffness of the plate (all element types and packages gave similar answers and were considered adequate) and the stress distribution around the hole. Overall the best results were obtained using the 9-noded elements and the 8-noded elements with full integration, although all element types gave similar answers for hoop and radial stresses around the pin. There were significant differences in the shear stress distributions calculated with the worst results obtained form the 4-noded elements.

References

Hollaway, L., Farhan, A and Gunn, M. J., (1987), Some dynamic analyses of a unit tetrahedral truss in both earthed and near free/free conditions, The Society of Engineers Journal, 78, No. 3, 33-60.

Romhi, A. (1990), Non linear stress analysis of a composite end cap connected to composite skeletal systems, PhD thesis, University of Surrey.

Romhi, A., Hollaway, L. and Gunn, M. J. (1986), An elastic and an elasto-plastic analysis of a unique joint for pultrusion composites, Composite Structures, 6, 197-239, Elsevier, England.

FINITE ELEMENT ANALYSIS OF COMPOSITE NAVAL STRUCTURES

J D McVEE and R S DOW

Defence Research Agency, Maritime Division
ARE, St Leonard's Hill
Dunfermline, Fife, KY11 5PW

An overview is presented of the application of finite element techniques to the structural analysis and design of composite naval structures carried out during the last twenty-five years (1)* at the Dunfermline site of the recently formed Defence Research Agency, Maritime Division. It includes a brief description of marine applications of laminated glass reinforced plastic (GRP) composites along with a short discussion of typical laminate properties, permissible stress levels and relevant failure mechanisms. Where applicable, other theoretical methods such as laminated orthotropic beam theory, folded plate techniques and axisymmetric shell stress and buckling analysis are also considered. The importance of theoretical results correlation with full and model scale test data is emphasised.

In particular, extensive use of several linear and non-linear proprietary finite element computer codes in the evolution (2, 3, 4) of three classes of GRP mine countermeasures vessels (MCMVs) developed by the Royal Navy during the last thirty years is very briefly described. Examples are given of primarily linear finite element analysis ranging from the determination of the global elastic response of large scale MCMV test sections, corrugated GRP hulls, and subsequent detailed examination of various GRP joint designs to the prediction of stress distributions in typical GRP interlaminar tensile strength specimens. Non-linear analysis concerned with compressive buckling of longitudinally stiffened panels (5), interactive buckling effects (6) and the initial buckling and post buckling of grillages subjected to combined shear and direct loading (7) are discussed.

Recent studies concerned with the potential application of hybrid GRP/steel superstructures are outlined. These consist of preliminary static elastic analysis (8) which indicate that fatigue failure associated with hull-superstructure interaction can be significantly reduced. Moreover, dynamic response analysis (9) of representative full-scale superstructure deck panels, and a corresponding three dimensional deckhouse module, have been correlated with experimental test data which demonstrates the capability of this form of construction to withstand external air blast loading.

With reference to submarines, a series of linear static elastic response and buckling analysis (10) performed on a sequence of models representing a full scale GRP deck casing test section subjected to proof load pressures are described which confirmed that failure was due to through thickness shear effects. Regarding the potential use of fibre reinforced polymer (FRP) components in the pressure hulls of submersibles (11), the theoretical performance of FRP is contrasted with that of steel and light alloy construction in two illustrative trial designs corresponding to a shallow water manned submersible and a deep water autonomous underwater vehicle. Axisymmetric thin and thick shell and three dimensional non-linear finite element discretisations necessary to obtain critical buckling pressures are highlighted.

Finally, some suggestions are made for code improvements including increased ease of data input, improved elements, more general stress-strain definitions, access to more general failure criteria and more user orientated visual pre and especially post processing capabilities.

* REFERENCES

1. Smith, C S. Design of Marine Structures in Composite Materials, Elsevier, London, 1990.

2. Smith, C S. Structural problems in the design of GRP ships, Proc. Symp. on GRP Ship Construction, RINA, London, 1972.

3. Chalmers, D W, Osborn, R J and Bunny, A. Hull Construction of MCMVs in the United Kingdom, Proc. Int. Symp. on Mine Warfare Vessels and Systems, RINA, London, 1984.

4. Mableson, A R and Smith, C S. The evolution of GRP hull construction for the Royal Navy's MCMVs, Maritime Defence, August 1988, p.293.

5. Smith, C S and Dow, R S. Compressive strength of longitudinally stiffened GRP panels, Proc. 3rd Int. Conf. on Composite Structures, Paisley; Elsevier, London, 1985.

6. Smith, C S and Dow, R S. Interactive buckling effects in stiffened FRP panels, Proc. 4th Int. Conf. on Composite Structures, Paisley; Elsevier, London, 1987.

7. Dow, R S and Smith, C S. Buckling and post buckling behaviour of stiffened composite panels under shear and combined loads. Int. Conf. on Practical Design in Shipbuilding (PRADS 89), Varna, October 1989.

8. Smith, C S and Chalmers, D W. Design of ship superstructures in fibre reinforced plastic, Trans RINA, 129, 1987, p.45.

9. Smith, C S and Murphy, P J. Response of hybrid GRP/steel superstructures to blast loading - theory and experiment, Advances in Marine Structures 2, Elsevier, London, 1991.

10. McVee, J D. Developments in finite element analysis of naval structures, Advances in Marine Structures, Elsevier, London, 1986.

11. Smith, C S. Design of submersible pressure hulls in composite materials, Marine Structures, 4, 2, 1991, p.141.

ABSTRACT

ANALYSIS OF COMPOSITE RACING YACHTS

Patrick Dallard
Ove Arup & Partners, Consulting Engineers
13, Fitzroy Street
London, W1P 6BQ

Tel: 071-465-2264

This presentation describes the evolution of a method for the analysis of composite racing yachts, along with the development of the required software. It is based on Arup experience in the design and analysis of the hull structure for the International Offshore Racing yacht 'Satquote British Defender' and an America's Cup Class yacht for an Australian syndicate. Most recently the techniques have been used to analyse Dennis Conner's latest America's Cup yacht 'Stars and Stripes'.

The presentation discusses:

- The significance of the hull shell shape to its structural behaviour.
- How a finite element model can be created from the available geometric data.
- How material input data can be manipulated.
- What load cases need to be considered and how the loading is applied to the model.
- The special post processing requirements posed by composites and what data reductions techniques can be used.

Some developments in finite element aided design for composite structures

by Georges Verchery
EMSE
42023 ST ETIENNE CEDEX 2
FRANCE

Presentation at the workshop
Finite element methods for composites
5-6 September 1991
Imperial College, London

This paper presents attempts and current developments in softwares for finite element aided design of composite structures. The principal objectives of these studies are to obtain methods and programs easy to use and capable to run on micro-computers.

The complexity of the behaviour of composite materials has led to more and more complex theories implemented in more and more complex special finite element methods. Presently, very sophisticated and precise capabilities are available for *analysis*. However, these softwares are less and less easy to use : the collection and the imput of the data (specially the materials data) are long or difficult, the selection of the adequate theory or the adequate elements (among so many available) may be a real challenge, the programs generally require large computers, finally the large number of numerical results do not receive easily a precise and useful physical meaning.

On the other hand, compared to these numerous, well-developed and complex methods for analysis, the *design* methods are poor. They mostly reduce to very specific methods (generally using a lot of algebra) or to optimization process : using some analysis method in each loop of their iterative course, these optimization programs may require huge computer facilities and long computation times.

From these considerations, it appears useful to develop simple and fast methods of computation. We retained the features described in the following.

1 - Re-analysis aided design : Fast analysis and re-analyses with finite elements constitute a possible way for improving design : instead of an automatic design program, the user has to analyse first, assess the results, decide for the structural changes, re-analyse the structure, assess the new results, and so on. A very fast finite element program is required for that. We developed various improvements in the finite element method to speed the computation. First, dimensional analysis and separation of the material and the geometric parameters have shown that pre-computed quantities can be used to form the stiffness matrices, which results in a faster formation of these matrices. Systematic use of multi-loads, in the initial analysis, considerably shortens the subsequent re-analyses for the cases obtained by combination of solutions. The flexibility method proved to be an efficient way for fast re-analysis with different boundary conditions. Finally, perturbation methods are presently investigated to take into account structural modifications in the flexibility matrices.

2 - **Two-step search :** Design of composite laminated or sandwich structures should include the choice of the nature of the laminas, and the determination of the stacking sequences. Consequently, the complete problem of optimization -or, more simply, of improvement- of design is material and structural. To simplify the approach, we introduce a two-step process. The first step has to find improved overall structural properties, while the second step consists in finding a material which can reproduce these properties. Two principal comments should be emphasized. First, this two-step process, while simpler, may generally provide a "less optimal" solution than a complete optimization (if it exists and can be effectively processed). Second, the material design for given structural properties is questionable : it is rather doutfull that one can obtain the wanted structural properties with the available basic laminas. However, we solved this difficulty by extending the range of search possibilities : it is possible to define, in a quantitative way, norms and deviations for anisotropic structural properties, so that, instead of solely searching for materials which give exactly the required structural properties, we can search for materials which give approximately these properties, within a definite relative deviation.

3 - **User friendly programs :** In addition to fast processing, already discussed, at least two requirements must be fulfilled in order to have user-friendly softwares. They must be interactive, specially with graphic facilities, at the input and output phases. They must run on largely available and easy-to-use computers. To meet these requirements, we retained micro-computers (IBM PC-AT or compatible, and MacIntosh II) and low cost work-stations. We also included in our programs (or plan to include in the programs currently in development) user-friendly facilities, such as use of mouse, pop-up menus, graphical windows, etc.., in the way made popular by the Apple softwares.

Two softwares have been developed on the above ideas and methods. The program FEAD-LASP (Finite Element Aided Design for Laminated And Sandwich Plates) was developed from parts of a rather classical finite element program for micro-computer. It is limited to rectangular geometries (composite and sandwich panels in bending and in-plane deformation) and pre-defined regular meshes, but takes into account completely general boundary and load conditions. Generality is also available for the material properties, which include bending, in-plane, bending-membrane coupling and transverse shear stiffnesses (extended laminated plate theory, i.e. classical laminated plate theory plus shear effects). A 16-node thick plate element with transverse shear, in which the materials properties of the laminate or sandwich are input using an equivalent material, was retained. A faster method for the formation and assembly of the stiffness matrices of the elements was derived and tested. It reduces the corresponding time very significantly, using precomputation and storage of dimensionless parts of the stiffness matrix elements. Several micro-computers and work-stations were used to test the numerical part : it was found that the most powerful personal computers (such as Apple Macintosh II and IBM PC-AT) as well as the low cost work-stations (such as Apollo DN-3000) are well suited for a fast numerical treatment, i.e. only a few minutes for the first analysis, and re-analyses in seconds. A user-friendly input part for the program was developed and makes possible the use of the program by designers with little or no knowledge of the internal structure and methods of the program.

The program FEAD-FLEX (Finite Element Aided Design by Flexibility method) is currently in development and test. It already includes in its features the fast re-analysis with varying the boundary conditions. The principle of the method is the following : taking advantage of the fact that the rigid body motions are the singularities of the equilibrium system of discrete elastic structures, a general solution can be expressed prior to any assignment of displacement boundary conditions, then the solution for specified load and boundary conditions can be determined by solving an associated small-size linear system. With this method, the re-analyses under various boundary conditions are made easier. Further, it can simplify the solutions for contact problems and elastic crack propagations problems, which otherwise need long iteration methods, and be useful for substructuring of large or repetitive structures. Finally, perturbation methods for the flexibility matrix are currently implemented in this software. They allow to take into account local modifications of the stiffness (due to geometric or material changes) without coming back to the beginning of the process, and consequently reduce the computation time. We plan to use this method for various problems in the mechanics of composite materials, such as damage propagation analysis, and form optimization.

Title FEM applied to a truck frame made of hybrid composites

Authors: Antonio Miravete, Pedro M. Bravo and Emilio Larrodé

Address: Centro Politecnico Superior de Ingenieros
María de Luna, 3
50015 Zaragoza (Spain)

ABSTRACT

Among the applications we are analyzing currently, the truck frame is the most interesting from the point of view of problems and successes of the application of finite element method to a composite structure.

The frame, which is the object of the present study, is composed of three substructures (Figure 1).

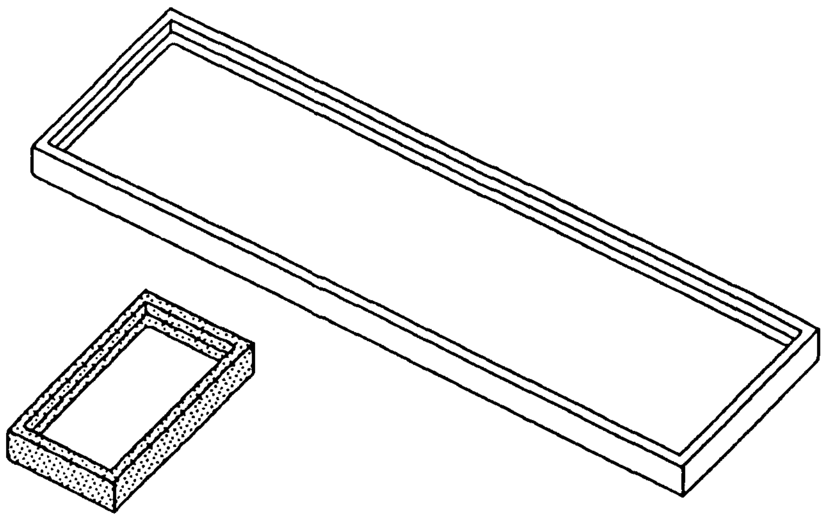


Figure 1. Scheme of the truck frame

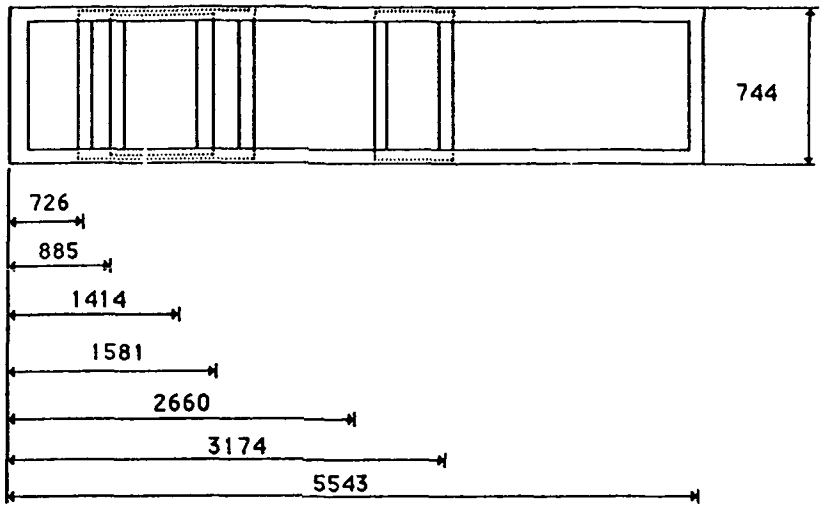


Figure 2. Dimensions of the truck frame in mm.

There are two problems :

* Stress analysis in the corners

* Stress analysis in the variable thickness areas

The variable thickness problem was solved by means of 8 node and linear displacement brick. But the corner was an interesting area to analyze.

Analysis

Since the corner is one of the most critical substructutres, the following testings have been carried out :

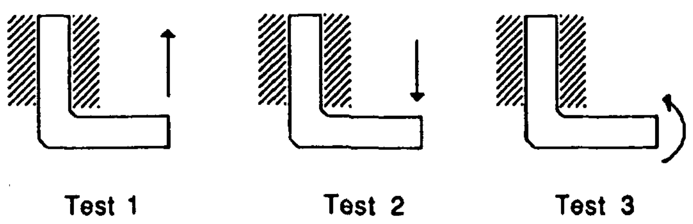


Figure 3. Scheme of the tests 1, 2 and 3.

The material used is :

Flanges : a unidirectional hybrid carbon fiber/epoxy resin (10%) and fiberglass/epoxi resin (90%) composite material.

Web : a unidirectional fiberglass/epoxy resin composite material.

In both cases, volumen fraction is 0.6.

The following ABAQUS fifteen finite elements have been used for the analysis or these three testings :

S4R	4 node, reduced integration, doubly curved shell with hourglass control.
S8R	8 node, reduced integration, doubly curved shell.
S4R5	4 node, reduced integration, doubly curved shell with hourglass control, using five degrees of freedom per node.
S8R5	8 node, reduced integration, doubly curved shell ,using five degrees of freedom per node.
S9R5	9 node, reduced integration, doubly curved shell, using five degrees of freedom per node.
C3D8	8 node, linear displacement brick.
C3D8R	8 node, linear displacement, reduced integration brick including hourglass control.
C3D8H	8 node, linear displacement, constant pressure brick.
C3D8RH	8 node, linear displacement, constant pressure, reduced integration brick including hourglass control.
C3D20	20 node, quadratic displacement brick.
C3D20R	20 node, quadratic displacement, reduced integration brick.
C3D20H	20 node, quadratic displacement, linear pressure brick.
C3D20RH	20 node, quadratic displacement, linear pressure, reduced integration brick.
C3D27	27 node, cubic displacement.
C3D27R	27 node, cubic displacement, reduced integration brick.

Results

Test 1. C3D27 element is the most accurate (4 %) though results from 20 and 27 node element are vey close to the experimental data. The rest of the elements give very poor results.

Test 2. C3D27 and C3D27R elements give good results (3 and 3.7 % error, respectively) and 20 node elements present errors around 10 %. C3D8(H) element error is 39 % and the rest of 8, 4 and 9 node elements present very poor results.

Test 3. Results from 20 and 27 node element are vey close to the experimental data (errors from 6 to 8 %). The rest of the elements give very poor results except C3D8(H), S4R5 and S8R5 (15 to 20 % error).

NUMERICAL PROBLEMS ENCOUNTERED IN NONLINEAR FINITE ELEMENT ANALYSIS OF FREE EDGE DELAMINATION

J.C.J. Schellekens
Delft University of Technology, Department of Civil Engineering /
TNO Building and Construction Research
P.O. Box 5048, 2600 GA Delft, Netherlands

ABSTRACT

A failure mode often encountered in structural composite laminates is delamination, a phenomenon which has great influence on the structural integrity. In the last two decades much research has concentrated on the complex mechanism of delamination. Due to the anisotropic material properties and the varying fibre orientations each ply in the laminate behaves differently. Large edge stresses are then necessary to preserve compatibility of the deformations. These large transverse normal and shear stresses are primarily responsible for the initiation of delamination. In the literature various analytical models have been developed for the treatment of free edge stress distributions, e.g. by Pagano. Furthermore procedures to predict delamination onset and growth based on the principle of virtual crack extension have been applied by Crossman, Wang and O'Brien amongst others. Because of the stress singularities that exist at the ply interfaces near the free edges and the resulting mesh-dependence it is commonly believed that the use of stress-based failure criteria does not produce relevant results as far as free edge delamination is concerned. However, Kim and Soni indicated that an average stress approach combined with an anisotropic failure criterion results in an accurate prediction of the onset of delamination. The introduction of the ply thickness in the determination of stresses is essential in their approach.

In this contribution a nonlinear finite element procedure is presented for the prediction of delamination onset and growth. The procedure accounts for thermal effects since we have to take into account the initial stresses in the laminate due to the forming process. Although the failure criterion is based on stresses it is shown that, when combined with a softening type of post-crack response, a stress-based failure criterion results in a mesh-objective calculation. The performance of the method is demonstrated by means of the analyses of

free edge delamination in various graphite-epoxy laminates and the numerical results are compared with experiments.

In the examples twelve-noded cubic generalised plane strain elements with three translational degrees-of-freedom in each node have been used. These elements, which are assumed to remain elastic during the loading process, give an accurate representation of the stress concentrations near the free edges without the need for extreme mesh refinement. In contrast to other continuum elements generalised plane strain elements are not loaded by nodal forces or prescribed displacements but by prescribed strains normal to the element plane. The individual plies are connected by cubic 3D line interface elements which are well suited for modelling the geometric discontinuity that arises during delamination, which can either be gradual (softening type behaviour) or perfectly brittle

In the lecture the emphasis will be put on the effects of mesh refinement and laminate thickness on the ultimate load capacity of the laminates. It will be demonstrated that the inclusion of interface elements in the finite element model results in a proper description of size-effects and mesh-objectivity due to the softening type of response after cracking.
Since the application of a conventional Gauss integration scheme for the assembly of the stiffness matrix of the interface elements may result in a poor element performance, the effect of other integration schemes on the element behaviour is investigated.
A major drawback of strain or load-controlled calculations is the fact that no limit points in the load deflection curve can be passed. Riks developed an 'arc-length' method to overcome these limitations. Herein the incremental load-factor is constrained by the norm of the incremental displacement vector. Although the arc-length control method has proved to be rather successful, it has been reported to fail in situations of highly localised failure. In these cases the displacement norm should be determined considering only the dominant degrees of freedom. Therefore a modified version of the standard arc-length method will be discussed which results in a stable solution process upon passing limit points in the structural response.

Thermal Analysis of Laminated Composites
- Element Development and Experience with MSC/NASTRAN

Raimund Rolfes

DLR, Institute for Structural Mechanics,
Flughafen, D-W3300 Braunschweig, Germany

Workshop: Finite Element Methods for Composites, September 5th-6th, 1991, London

1 INTRODUCTION

Thin-walled structures from laminated composites are supposed to play an increasing role in future, especially for aerospace applications. The thermal loads will in many cases decisively influence the design. For example, this applies to the load-carrying structure of spaceplanes, large orbital structures or even primary components of tomorrow's commercial airliners.

2 THERMAL ANALYSIS WITH MSC/NASTRAN, VERSION 66A

Subsequently, the most important capabilities required from a Finite-Element thermal analysis code are specified and compared with the features of MSC/NASTRAN's version 66A [1]. Some of the following topics are specific for composites, others refer to thermal analysis in general.

- Suitable elements taking into account the thermal anisotropy and inhomogeneity of laminated composites

  MSC/NASTRAN provides 2D- and 3D-elements with thermal anisotropy. However, there is no element for laminates. This shortcoming has been tackled by our elements QUADTL and QUADQL, which are described in chapter 3 of this paper.

- Boundary conditions and loads

  They should be as flexible as possible, e.g. they should include time and temperature dependent convection, temperature dependent emissivity, etc.
  MSC/NASTRAN has full temperature but no time dependent convection. The emissivity can only be made temperature dependent in transient analysis, not in steady state calculations.

- Nonlinearities

  They occur in most thermal problems due to radiative boundary conditions and temperature dependent thermophysical properties. Especially, the latter is important for composites, since the conductivity in fibre direction of an ordinary carbon-epoxy increases by about 60% between 30°C and 120°C, while alumina (99%) has a nearly constant conductivity in the range of 0°C to 500°C [2].
  MSC/NASTRAN can handle radiative boundary conditions, while temperature dependent conductivities in transient analysis are difficult to incorporate and temperature dependent heat capacities are impossible.

- View factors for radiation exchange

  This is an important issue for many aerospace structures, especially satellites. MSC/NASTRAN has a module for automatic calculation of these factors. However, it works very unsatisfactory at present.

- User friendliness

  Especially modelling of boundary conditions should be easy.
  MSC/NASTRAN has special heat boundary elements for convective and radiative boundary conditions. They are supported by the interface program PATNAS (interface to PATRAN) [3]. However, modelling of temperature boundary conditions in transient analysis costs a lot of effort . Another inconvenience is the lack of an automatic time stepping algorithm in transient problems.

- Interfaces to pre- and postprocessors

  Items like heat flux loads and radiative as well as convective boundary conditions were very poorely supported up to version 2.2 of PATNAS. This has been improved recently within version 3.0 (January 1991) [3],[4].

According to MacNeal-Schwendler the shortcomings mentioned above will be eliminated in version 68, which is supposed to be released in 1993.

3 THERMAL ANALYSIS OF LAMINATED COMPOSITES VIA THE FINITE ELEMENTS QUADTL AND QUADQL

The thermal analysis of composites becomes an anisotropic and inhomogeneous problem, if fibres and matrix exhibit different thermal conductivities. Especially, this applies to high- modulus carbon fibre reinforced plastics, where the degree of thermal anisotropy can reach 180. This is the ratio of the principal conductivities of a lamina in fibre direction and perpendicular to it (k_I/k_{II}).

A Thermal Lamination Theory (TLT) applying to plate and shell structures with high degree of thermal anisotropy has been developed [5],[6]. It assumes a linear temperature distribution over the thickness and can be regarded as the thermal analogue to the Classical Lamination Theory (CLT). The stacking sequence of the laminate is taken into account by introducing "moments of heat flux" as equivalent forces. The isoparametric quadrilateral finite element QUADTL based on the TLT has eight degrees of freedom - one at each node for the temperature of the laminate's reference surface and another for the temperature gradient in thickness direction. Steady state calculations of plates, subjected to a very concentrated heat flux, showed a favourable agreement with fully three-dimensional calculations, while the computing time was reduced by two orders of magnitude. However, the heat flux calculated from the derivatives of the temperature field is approximated worse. Moreover the linear temperature distribution over the thickness might be to rough in the case of transient heat conduction.

That's why a higher order theory, called the Quadratic Lamination Theory, has been established [7]. The finite element QUADQL is related to the new theory and has twelve degrees of freedom. Numerical examples provided excellent temperature and heat flux results for steady state as well as transient problems.

4 REFERENCES

[1] The MacNeal-Schwendler Corporation (ed. W.H. Booth):
MSC/NASTRAN Version 65, Handbook for Thermal Analysis (1986)
(No difference to version 66A)

[2] Rolfes, R.:
Wärmeleitzahlen von UD-Laminaten aus CFK - Berechnung und Messung, Interner Bericht 131-91/12, DLR (1991)

[3] PDA-Engineering (ed.):
PAT/MSC-NASTRAN Application Interface, Release 2.2, Costa Mesa, Ca., U.S.A. (1989)

[4] PDA-Engineering (ed.):
PAT/MSC-NASTRAN Application Interface, Release 3.0, Costa Mesa, Ca., U.S.A. (1991)

[5] Rolfes, R.:
A New Finite Element for Thermal Analysis of Laminated Composite Plates, Proceedings of the 17th MSC European User's Conference, Lect. No.30, Sept. 25-28 (1990)

[6] Rolfes, R.:
Efficient Thermal Analysis of Anisotropic Composite Plates Using New Finite Elements, In: Füller, J.; Grüninger, G.; Schulte, K.; Bunsell, A.R.; Massiah, A.: Developments in the Science and Technology of Composite Materials, Proceedings of the Fourth European Conference on Composite Materials (ECCM4), Stuttgart, pp. 743-748 (1990)

[7] Rolfes, R.:
Higher Order Theory and Finite Element for Heat Conduction in Composites, in: Lewis, R.W., Chin, J.H., Homsy, G.M.: Numerical Methods in Thermal Problems, Vol.VII, Proceedings of the Seventh International Conference, Stanford, U.S.A., pp. 880-889 (1991)

Simulation of the Thermoforming Process of Continuous Fibre-Reinforced Thermoplastic Composites Using MARC

R. Scherer, Technical University Hamburg-Harburg, Germany

RESEARCH PROJECT

Thermoforming is one of the most promising techniques to manufacture components made of continuous fibre reinforced thermoplastic composites. The main feature is the molten state of the thermoplastic matrix during forming. The laminate can be modelled as an alternating sequence of fibre rich layers and resin interlayers, where the height of the interlayer is approximately 20 μm for a carbon fibre/polypropylene composite. The individual fibre rich layers have to slip onto each other to accomodate onto a rigid tool.

"Ply-Pull-Out" experiments characterized the "interply-slip" behaviour in dependency on the interlayer height, forming speed, processing temperature, pressure applied normal to slip direction and lay-up configuration. These experiments represent a one-dimensional characterization of the interply-slip effect. The measured data deal as input parameters for a Finite Element Analyses of a thermoforming process.
First, the material law was defined by two different ways with MARC, and the "ply-pull-out"-experiments were simulated, whereby the material law is nonlinear and rate dependent.
Two-dimensional, 8-noded plane strain elements were used for the modelling studies.

This material nonlinear behaviour is superimposed by a geometrical nonlinear mode in two-dimensional modelling studies. Initially flat laminates were thermoformed into a 90°-angle and a "top-hat"-section in between two rigid dies. The deformation behaviour of two-layered laminates were investigated so far, and the resin interlayer was reduced to a contact surface in between deformable elements. A user-defined subroutine, written in "Fortran", defined the properties of this interlayer. The main advantage of a computer simulation is the possiblity to vary the parameters of

the interlayers so that the stress transfer from the interlayers into the fibre rich layers can be visualized. This was not possible during thermoforming experiments which have been carried out simultaneously. The more the viscosity, namely, the "inner friction" of the resin interlayers is increased, the higher the tendency towards buckling of plies is pronounced.
Both, the incremental deformation behaviour during experimental and computer simulation studies lead to the same results which encourages to say that an appropriate material model for the interlayer was found.

Three-dimensional modelling studies are currently in process. A thermoforming process of a hemispherical part in between two rigid dies is modelled with shell elements. The material properties of the fibre rich layers are strongly anisotropic. On the one hand side, the fibres dominate the elastic response against deflection. On the other side, the matrix possesses a diminishing material response in the direction perpendicular to the fibres.

DRAWBACKS DURING FINITE ELEMENT ANALYSIS

MARC is able to simulate material and geometrical nonlinear behaviour as described above. The main drawbacks result from a limited usage of different program facilities simultaneously. For example, the material nonlinear description requires a time incrementation for the calculation of rate depenedent properties and the geometrical nonlinear analysis requires a time incrementation for the movement of the dies. Unfortunately, both analyses cannot be carried out at the same time. Therefore, it is not possible with MARC to calculate a forming process of a rate-dependent visco-plastic material.
A general disadvantage of some FE-programs is the "suspicious" definition of convergence criteria for a nonlinear iterative calculation. Additionally it is recommendable that the size of increments should be automatically adapted to the current program in the ideal case. If the increments are manually defined, both small and large increments can lead to significant different results. Of course, a rough incrementation should be used to save computing time, but it often leads to a large number of recycles

within one increment. Therefore, user manuals should outline the influence of the different paramerers in a nonlinear analysis in more detail.

Abstract submitted for oral presentation on the workshop "Finite Element Methods for Composites", 5/6 September 1991, Imperial College of Science, Technology and Medicine, Centre for Composite Materials, London, England.

60

The Prediction of Three-Dimensional Properties of Composite Laminates Using the Finite Element Analysis Method

F. J. GUILD
Department of Materials, Queen Mary & Westfield College, Mile End Road, London E1 4NS, UK

J. D. STRAUGHAN and N. ZAHLAN
ICI plc, Wilton Materials Research Centre, PO Box 90, Wilton, Cleveland TS6 8JE, UK

ABSTRACT

A procedure for predicting the properties of composite laminates in three dimensions is presented. Analysis has been carried out using two commercially available finite element analysis codes. Constraint conditions have been applied to the model to simulate the conditions of classical laminate theory and the in-plane predictions of both procedures are compared. Mesh refinement in both the thickness and in-plane dimensions has been investigated and optimised. The resulting procedure will predict out-of-plane as well as in-plane properties of composite laminates using economically small models.

1 INTRODUCTION

The first use of advanced continuous fibre reinforced composites, in the early 1960s, was in direct substitution for metallic panels on aircraft.[1] At this initial stage the laminates were mechanically fastened to a metallic framework. The great reduction in weight was sufficient to justify the high cost of the materials and the manually intensive fabrication procedures employed.

In this plate-like form it was justifiable to assume the simplifications associated with the engineering mechanics approach applied to thin plate behaviour. Essentially this approach neglects through thickness behaviour of the material including shear stresses. In addition, linear elastic behaviour was assumed since the materials were based on thermoset-based matrix systems with limited strain capacity. Consequently a design methodology emerged which is heavily based on these assumptions. This allowed use of the existing metal-based analysis procedures, after minor modifications to allow for orthotropy of material properties, for the analysis of composite structures.

This methodology relies on measured in-plane mechanical properties of the basic composite materials. For a unidirectional composite these are: two elastic moduli, one shear modulus and one lateral contraction ratio, Poisson's ratio. Procedures based on classical laminate theory (CLT)[1] are employed for prediction of in-plane elastic properties of composite laminates containing plies oriented at any angle (multi-angle laminates). Effective elastic properties derived from CLT are then used in engineering mechanics approaches which enable the analysis of simple plate-like structures subjected to selected idealised loading modes. The elastic properties derived from CLT are also used in modified numerical analysis tools such as the finite element analysis (FEA) method; this enables the analysis of structures of complex geometry.

In recent years many developments and changes have taken place. Composite materials with significant toughness have been introduced, first in thermoplastic-based form[2,3] and later thermoset-based.[4,5] These materials have a significant resistance to delamination, the traditional weakness of composites. The ability to join composites to other materials and to one another has improved significantly through the development of better adhesives for all composites, and bonding and welding techniques suitable for thermoplastic-based composites.[6] Fabrication techniques capable of achieving rapid forming of complex shapes, such as rubber block stamping, have been developed for thermoplastic composites.[7] Methods applicable to thermoset-based composites are also being developed. Finally and significantly, experience with composite materials indicates that the majority of failures which occur below the design load level are due to through thickness stresses.[8]

In view of these developments, composite structures are no longer restricted to simple plate form attached to metallic frames. Plate-like laminates can be formed into complex shapes with intricate integral supporting frameworks. Composites are joined together to form all-composite structures. In addition, modern composite materials are used to build three-dimensional solid components.[9,10]

The use of existing analysis methods and their assumptions for present-day materials and applications must be questioned. Analysis procedures and tools which account for through thickness behaviour of the materials are needed. Investigators have proposed procedures based on three-dimensional constitutive equations[11] and the longwave approach.[12] We have proposed a numerical approach based on the finite element analysis (FEA) method which allows prediction of both in-plane and out-of-plane properties of composite laminates.[13] In this paper we extend the approach and consider the effect of finite element mesh density on the predicted values. We compare the results to those produced using CLT, and propose a cost-efficient procedure for predicting through thickness properties as well as in-plane properties without the restrictions of plate behaviour. This procedure can be used for the investigation of the effects of inclusion of interlayers and laminae of different thicknesses in a composite, as well as examination of detailed stress distribution within the laminate.

2 MATERIAL

The material studied is a multi-angle laminate of continuous carbon fibre reinforced thermoplastic composite composed of semi-crystalline polyetheretherketone (PEEK) thermoplastic polymer matrix reinforced with 61% by volume of continuous Hercules AS4 carbon fibres. A procedure for determining the properties of the basic unidirectional material has been presented in Ref. 14. Measured properties inherently include an amount of uncertainty. An analysis of the sensitivity of the present predictive technique to the accuracy of the input data has been carried out and will be published elsewhere. For the present modelling a consistent set of input data, shown in Table 1, has been used. The conventional axes nomenclature is used, namely direction 1 being the fibre direction and directions 2 and 3

TABLE 1
Material properties

E_1 (GPa)	E_2 (GPa)	G_{12} (GPa)	G_{23} (GPa)	ν_{12}	ν_{23}
130·0	9·7	5·51	3·59	0·30	0·35

being the two orthogonal directions, in-plane and out-of-plane respectively. The lamina is assumed to exhibit transverse isotropy, that is the properties in directions 2 and 3 are assumed to be identical. The results presented here are for the analysis of a quasi-isotropic laminate $[+45/0/-45/90]_s$. As described below, the method of analysis used is suitable for any symmetric balanced laminate.

3 FINITE ELEMENT ANALYSIS

Finite element modelling has been carried out using two commercially available general-purpose codes, LUSAS and ABAQUS. Most analyses use 20-noded isoparametric, three-dimensional solid elements.[15] Two analyses, for comparison with the results from CLT, were carried out using eight-noded elements. Orthotropic material properties were used; all material behaviour was assumed to be linear elastic.

The development of the finite element model is shown in Fig. 1. The analysis is for an infinite laminate, which is represented by a block as shown. Since the laminate is balanced and symmetric, it exhibits no coupling of overall deformation either in-plane to out-of-plane or axial to shear. Since the laminate is represented by the block, the deformation of the representative block is assumed to be of the same form.

The representative block has three planes of symmetry, shown by the dashed lines in Fig. 1. Thus the deformation of the overall block can be analysed via the deformation of an $\frac{1}{8}$ of it, the cuboid ABCD$\alpha\beta\gamma\delta$. The deformation of ABCD$\alpha\beta\gamma\delta$ for the imposition of in-plane stress, in the y-

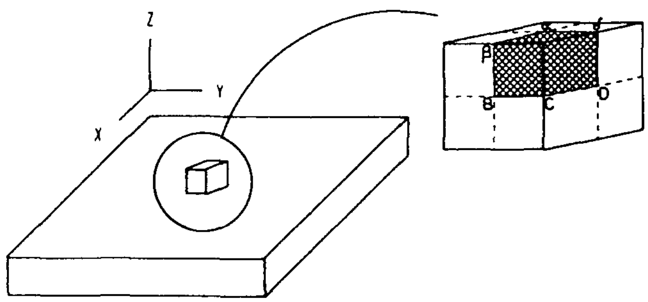


FIG. 1. Development of the finite element model.

direction, is shown in Fig. 2. The stress is applied via a prescribed displacement to the nodes of the face CDδγ. The planes ABβα and ADδα are restrained to remain in their original planes from the symmetry shown in Fig. 1. Constraint equations are applied to the nodes on planes BCγβ and αβγδ, forcing them to remain parallel to their original directions. Thus the deformation of a balanced symmetric laminate is imposed on the representative block.

Results from finite element analysis include reactions to earth, R, at surface nodes, and the displacement of all nodes, Δ. The elastic constants for the in-plane loading in the y-direction, shown in Fig. 2, are calculated from the finite element results via

$$E_y = \frac{R_y/(l_x \cdot l_z)}{\Delta_y/l_y} \qquad \nu_{yx} = -\frac{\Delta_x/l_x}{\Delta_y/l_y} \qquad \nu_{yz} = -\frac{\Delta_z/l_z}{\Delta_y/l_y}$$

The elastic constants from the other loading cases are calculated from the reactions and displacements from the respective finite element analyses by analogous calculations.

The finite element grids varied in the number of elements both for the out-of-plane, z-direction, and the in-plane directions, x and y (Fig. 2). The shape of the grid in-plane was always square, that is $l_x = l_y$ (Fig. 2). Analyses were carried out using cubic elements, and with longer aspect ratio elements

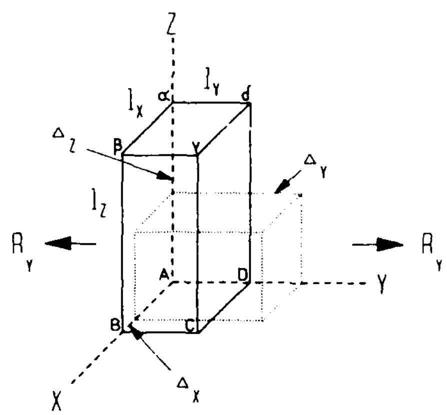


FIG. 2 In-plane loading of the finite element model.

TABLE 2
Finite element grids

Number of elements (in-plane)	Shape	Length/thickness	Package used
1 × 1	Cubic	0·083 33	LUSAS and ABAQUS
2 × 2	Cubic	0·166 7	LUSAS
5 × 5	Cubic	0·416 7	LUSAS and ABAQUS
5 × 5	Long	2·083	ABAQUS
10 × 10	Cubic	0·833 3	ABAQUS
10 × 10	Long	4·167	ABAQUS
15 × 15	Cubic	1·25	ABAQUS
15 × 15	Long	6·25	ABAQUS

with the two in-plane lengths being five times the out-of-plane length, so the ratio l_x/l_z is increased. This ratio is designated the length/thickness ratio, l/t. Finite element analyses were carried out for various column heights with a single element in-plane; as described below, these results allowed deduction of the number of elements required per lamina in the thickness direction. Further analyses were carried out, for the given column height, for various numbers of elements in-plane up to 15 × 15 elements. The grids analysed using 20-noded elements and 12 elements out-of-plane, that is thickness, are shown in Table 2.

4 RESULTS

4.1 Effect of Through Thickness Density

The effect of through thickness density was investigated by the analysis of a column containing a single cubic element in-plane. The finite element package used was LUSAS, using 20-noded elements. The laminate was analysed using grids of 8, 12 and 24 elements, that is with 2, 3 and 6 elements per lamina respectively. As expected, the results for the two in-plane directions, for values of both Young's modulus, E, and Poisson's ratio, ν, were identical. The results are summarised in Table 3. These results show that three elements per lamina allow accurate prediction of the through thickness modulus, E_z; the value is identical to that obtained using six elements per lamina. There is a small but measurable discrepancy in the values of Poisson's ratio, ν_{zx}; this discrepancy is less than 0·3% and is therefore considered insignificant. For the prediction of in-plane stiffness the results in Table 3 show that two elements per lamina may be sufficient, although this could lead to some discrepancy, around 0·8%, in the

TABLE 3
Effect of column height

In-plane, xx stress elements per lamina	E_x (GPa)	ν_{xy}	ν_{xz}
2	47·77	0·218 5	0·295 9
3	47·70	0·216 7	0·296 5
Out-of-plane, zz stress elements per lamina	E_z **(GPa)**	ν_{zx}	
2	10·12	0·062 58	
3	10·65	0·066 18	
6	10·65	0·066 37	

predicted value of the in-plane Poisson's ratio, ν_{xy}. It was concluded that the present analysis should proceed using three elements per lamina. All results presented below used grids with 12 elements in the thickness direction, the z-direction, representing four composite plies.

4.2 Comparison of Results from Finite Element Analysis and Classical Laminate Theory

The constraints of classical laminate theory (CLT) can be imposed in finite element analysis using a single column, that is one cubic element in-plane, of eight-noded elements. The allowable deformation of such a column of these elements imposes the condition of zero through thickness stress, that is the condition of plane stress assumed in CLT.

TABLE 4
Comparison of finite element and CLT results

In-plane, xx stress	E_x (GPa)	ν_{xy}	ν_{xz}
CLT	51·08	0·303	
ABAQUS	51·08	0·302 7	0·267 3
LUSAS	51·08	0·302 7	0·267 4
Out-of-plane, zz stress	E_z **(GPa)**	ν_{zx}	
ABAQUS	10·73	0·056 15	
LUSAS	10·73	0·056 16	

The results for classical laminate theory were calculated using COMLAN, an implementation of classical laminate theory developed in-house (ICI) using the LOTUS 123 spreadsheet. The finite element results were predicted using both packages, LUSAS and ABAQUS. The results are shown in Table 4. These results show complete agreement between all methods of prediction. Classical laminate theory cannot, by definition, predict any results out-of-plane, the z-direction. Finite element analysis using the same condition, plane stress, as CLT, predicts identical in-plane results to CLT, and also predicts out-of-plane results for this plane stress condition.

4.3 Results from Larger Finite Element Grids

Finite element analyses were carried out for larger size grids, using both cubic elements and elements with longer aspect ratios, as shown in Table 2. The predictions for the application of in-plane stress are shown in Figs 3–5.

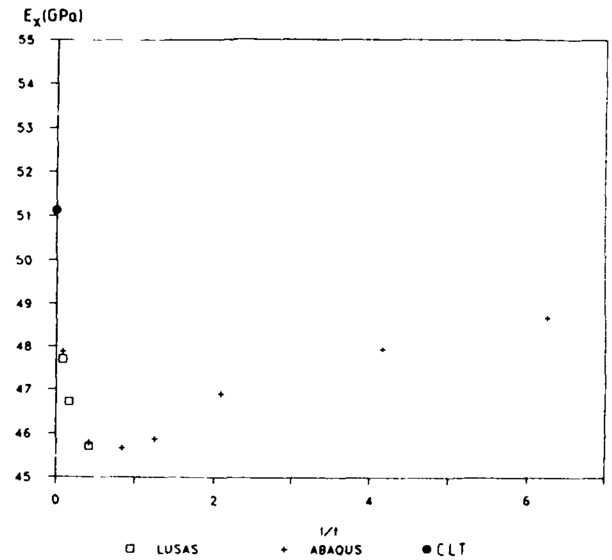


FIG. 3. Variation of the in-plane Young's modulus, E_x, with length/thickness ratio, l/t

All results are plotted against the ratio of the in-plane length to thickness, l/t. The value of this ratio indicates the relative influences of the edge effect on the result. For comparison the graphs also include the predictions assuming plane stress, described in Section 4.2.

The results show excellent agreement for the two packages used; the maximum discrepancy occurs for the value of the in-plane Poisson's ratio, ν_{xy}, predicted using a single cubic column; the discrepancy is 2%. The results from the different grids all lie on smooth curves. These curves are discussed in detail in Section 5.

The application of out-of-plane stress to the different grids produced insignificant variation in the predicted value of E_z; the predicted value varied between 10·60 and 10·67 GPa within the range of the length/thickness ratio shown in Figs 3-5. Thus prediction of the out-of-plane modulus is independent of the value of the length/thickness ratio and is not significantly different from the value predicted for plane stress conditions.

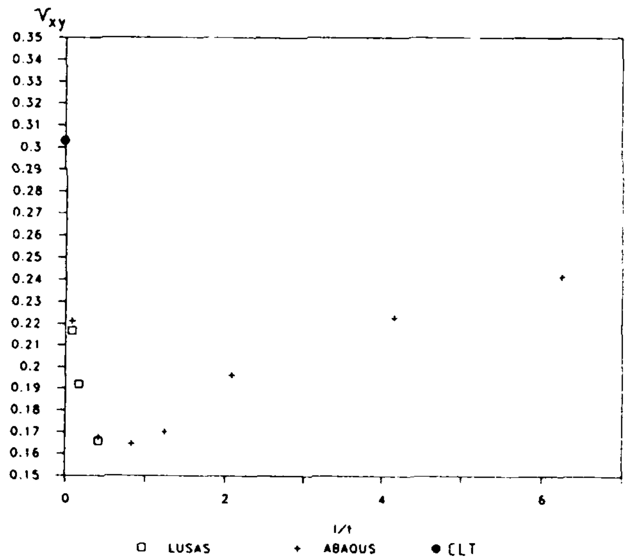


FIG. 4. Variation of the in-plane Poisson's ratio, ν_{xy}, with length/thickness ratio, l/t

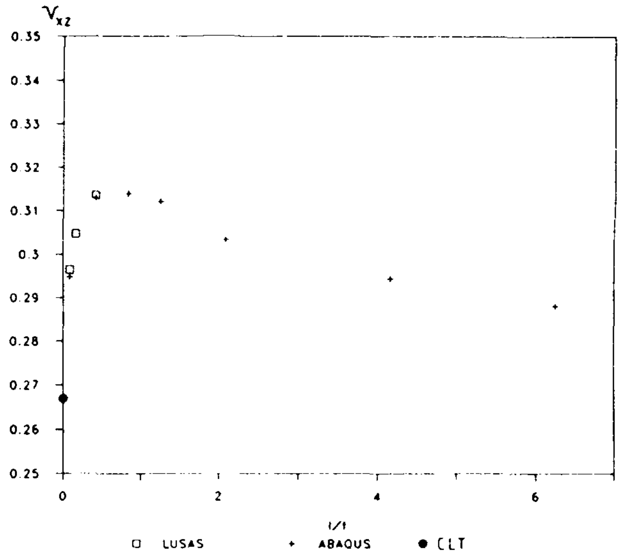


FIG. 5. Variation of Poisson's ratio, ν_{xz}, with length thickness ratio, l/t

10·73 GPa, shown in Table 4. The prediction of ν_{zx} is, however, dependent on the ratio; the variation is shown in Fig. 6. The value obtained assuming plane stress is also included for comparison. These results are similar to the results for ν_{xz} (Fig. 5), with all results on the same curve. This trend is discussed in detail in Section 5.

5 DISCUSSION

The results shown in Figs 3–6 may all be described in a single fashion. The results from the different grids may be plotted on single curves. In other words, the value predicted is related to the length thickness ratio of the grid. The value of this ratio represents the relative influence of the edge effects on the predicted result. Assuming that the size of the edge effect region is proportional to the laminate thickness, then as the length/thickness ratio

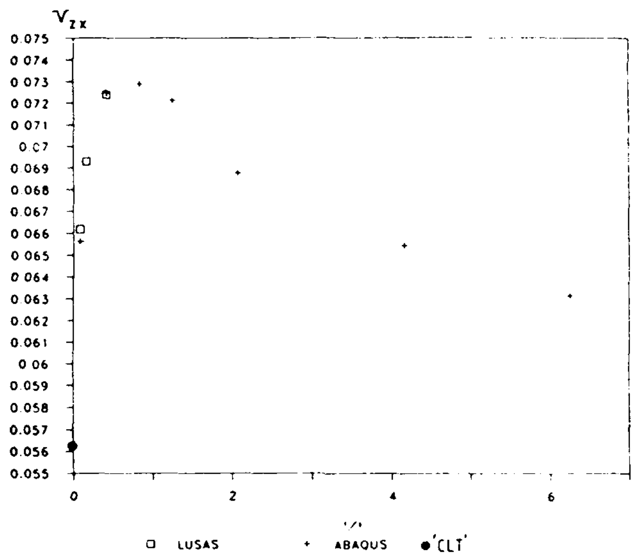


FIG. 6. *Variation of Poisson's ratio, ν_{zx}, with length thickness ratio, l/t.*

increases the influence of the edge region diminishes. Furthermore, the material within the edge region is less constrained than that in the interior region of the laminate, and hence apparently less stiff. As described previously, the overall distortion of the grid has been set for a balanced symmetric laminate, allowing no overall non-uniform distortion, as shown in Fig. 2. However, it is clear that distortions must occur within the laminate at the interfaces between laminae oriented in different directions. The mathematical formulation of a single column of eight-noded elements allows no such distortions to occur. The amount of permitted distortion increases as the model compliance is increased until a value of l/t is reached where the entire model represents the edge effect region. Beyond this value of l/t further increases in the model size represent increased interior region and hence increased stiffness.

The curves for the in-plane modulus, E_x, and the in-plane Poisson's ratio, ν_{xy} (Figs 3 and 4), follow similar trends. The maximum value is predicted using the assumptions of plane stress; the prediction is a minimum for a length/thickness ratio of about 1, and then increases. Both curves tend towards a constant value which lies between the value predicted for plane stress and the value predicted for the single column of cubic 20-noded elements. Thus the constraints imposed by plane stress, allowing no distortion between the laminae, increases the apparent stiffness and the overall lateral contraction; this is the result expected intuitively. Allowing inter-laminae distortions to occur produces a rapid reduction in the apparent stiffness and lateral contractions, reaching a minimum value as the in-plane thickness equals the thickness. At this stage the distortion is that of an edge region. As the l/t ratio is increased, the additional deformation is that of a more constrained interior region, thereby increasing the total effective stiffness to a value between that predicted for plane stress and the smallest length/thickness ratio, the single cubic column of 20-noded elements.

The results for the two values of Poisson's ratio relating displacements in-plane and out-of-plane, ν_{xz} and ν_{zx} (Figs 5 and 6), show similar trends. The predicted values are minima for the condition of plane stress, when no internal distortion is allowed. The lack of distortion would decrease the lateral contractions in one plane arising from the application of stress in the other plane. The predicted values reach maximum values at length/thickness ratios of about 1, and then decrease with the predicted value at an infinite length/thickness ratio apparently lying between the value for plane stress and the value predicted for the single column of 20-noded cubic elements. An analogous argument as used above explains this behaviour.

Conducting the complete series of models permits accurate evaluation of the mechanical properties. However, the models of the high l/t ratio are costly. Therefore, in the interest of economy, it is useful to establish two bounds within which the predicted properties will lie. One bound is obtained using the assumptions of classical laminate theory, and the other bound is obtained from finite element analysis of a single column of 20-noded cubic elements. However, the through thickness Young's modulus, E_z, may be obtained accurately from the result using the assumptions of classical laminate theory, that is the finite element analysis of a single cubic column of eight-noded elements. The results for the laminate considered here are summarised in Table 5. The predicted values have been derived from an examination of the curves shown in Figs 3–6.

The bounds shown in Table 5 are generally reasonably narrow. The exception is the in-plane Poisson's ratio, ν_{xy}; the variation between the bounds is more than 30%. This behaviour is not surprising since this

TABLE 5

Predicted elastic constant for [+45/0/−45/90]$_s$ quasi-isotropic laminate

	Lower bound	Predicted value	Upper bound
E_x (GPa)	47·8 via 20-noded	49	51·1 via 8-noded or CLT
E_z (GPa)		10·7 via all analyses	
ν_{xy}	0·219 via 20-noded	0·25	0·303 via 8-noded or CLT
ν_{xz}	0·267 via 8-noded	0·28	0·295 via 20-noded
ν_{zx}	0·0562 via 8-noded	0·062	0·0660 via 20-noded

mechanical property is expected to be highly sensitive to inter-laminae distortions. A more accurate value for this parameter could only be obtained via finite element analysis of a large grid with long aspect ratio elements. Reasonable bounds for all other parameters may be predicted using the single columns as shown in Table 5.

The predicted value of the in-plane modulus, 49 GPa, compares well with the measured value of 48·6 GPa.[14] The measured value of the in-plane Poisson's ratio, ν_{xy}, is 0·3;[14] this experimental value is within the bounds shown in Table 5 but not in agreement with the predicted value. This anomaly is the subject of future work.

6 CONCLUDING REMARKS

The three-dimensional mechanical properties of a multi-angle composite laminate have been predicted using a FEA-based procedure. Bounds for elastic constants of a quasi-isotropic laminate have been derived using economically small finite element grids. Some results from classical laminate theory have been shown to be usable. This procedure should be applicable to balanced symmetric laminates. The method can now be extended to the prediction of the elastic constants in shear, and the elastic constants of other laminates.

ACKNOWLEDGEMENT

The authors gratefully acknowledge valuable discussions with Mr D Hitchings, Imperial College.

REFERENCES

1. Jones, R. M., *Mechanics of Composite Materials*. Scripta Book Company, Washington, DC, 1975.
2. Cogswell, F. N. and Leach, D. C., Continuous fibre reinforced thermoplastics: a change in the rules for technology. *Plastics & Rubber Processing & Applications*, **4** (1984) 271–6.
3. ICI Fiberite, Aromatic polymer composite. APC-2 product data sheets, 1986.
4. Sefton, M. S., McGrail, P. T., Eustace, P., Chisholm, M., Carter, J. T., Almen, G., MacKenzie, P. D. and Choate, M., The use of thermoplastic polymers to improve the toughness of cross-linked networks. *Proc. 4th International Conf. on Cross-Linked Polymers*, 1989.
5. Almen, G., Maskell, R. K., Malhotra, V., Sefton, M. S., McGrail, P. J. and Wilkinson, S. P., Semi-IPN matrix systems for composite aircraft primary structures. *Proc. 33rd Nat. SAMPE Symposium & Exhibition*, 1988, pp. 979–89.
6. Cantwell, W. J., Davies, P., Jar, P. Y., Bourban, P. E. and Kausch, H. H., Joining and repair of carbon fibre composites. 11th Int. European Chapter of SAMPE Conf., Basel, 1990.
7. Cattanach, J. B. and Cogswell, F. N., Processing with aromatic polymer composites. In *Developments in Reinforced Plastics*–5, ed. G. Pritchard. Elsevier Applied Science Publishers, London, 1986, pp. 1–38.
8. Workshop on Micromodelling for Structural Composites, SERC, London, 1990.
9. Gehring, I. F., Carbon fibre polymer composite materials in weaving machinery engineering—examples of practical applications. *Melliand Textilberichte*, **70** (1989) 413–18.
10. ICI and Williams work on thermoplastic parts. *Advanced Composite Engineering* (March 1990).
11. Murthy, P. L. N. and Chamis, C. C., ICAN: Integrated Composites Analyzer. *Journal of Composites Technology & Research*, **8**(1) (1986) 8–17.
12. Sun, C. T. and Li, S., Three-dimensional effective elastic constants for thick laminates. *Journal of Composite Materials*, **22** (1988) 629–39.
13. Zahlan, N. and Guild, F. J., Computer aided design of thermoplastic composite structures. *Proc. 7th Int. Conf. on Composite Materials*, ICCM-VII, China, 1989.
14. Moore, D. R., Robinson, I. M., Zahlan, N. and Guild, F. J., Mechanical properties for design and analysis of carbon fibre reinforced PEEK composite structures. *Move into the 90's*, ed. S. Benson, T. Cook, E. Trewin and R. M. Turner. *Proc. 10th Int. European Chapter Conf. of SAMPE*, Birmingham, UK, July 1989. Elsevier, Amsterdam, 1989, pp. 167–76.
15. Cook, R. D., *Concepts and Applications of Finite Element Analysis*. John Wiley, New York, 1981.

IMPERIAL COLLEGE CENTRE FOR COMPOSITE MATERIALS
WORKSHOP: FINITE ELEMENT METHODS FOR COMPOSITES, 5TH & 6TH SEPTEMBER 1991.

CHAIRMAN

Prof I.C. Taig
Springfield, 9 Diglake
Tilstock
Whitchurch, Shropshire SY13 3NP

CENTRE DIRECTOR

Mr. F.L. Matthews,
Composites Centre.

SPEAKERS

Dr W.S. Arnold
Paisley College of Technology
High Street
Paisley PA1 2BE
Scotland

Prof F-K Chang
Aeronautics & Astronautics Dept
Stanford University
California 94305-4035
U.S.A.

Mr M.J. Gunn
Civil Engineering Dept
University of Surrey
Guildford
Surrey GU2 5XH

Mr J.D. McVee
Structural Analysis MTMS Dept
Admiralty Research Establishment
St. Leonard's Hill
Dunfermline, Fife
Scotland KY11 5PW

Dr A. Miravete
Dpto de Ingenieria Mecanica
Universidad de Zaragoza
Maria de Luna 3
50015 Zaragoza, Spain

Prof O.O. Ochoa
Mechanical Engineering Dept
Texas A & M University
College Station
Texas 77843-3123, U.S.A.

Dr R. Rolfes
DLR Institut für Strukturmechanik
Postfach 3267
D-3300 Braunschweig, Germany

Dr R. Scherer
Polymer Composites Group
Technische Universität Hamburg-Harburg
P O Box 90 10 52
D-2100 Hamburg 90, Germany

Dr H. Schellekens
TU Delft
Faculty of Civil Engineering
Stevinweg 12628 GA Delft
The Netherlands

Mr J.H. Underwood
US Army Armament Research
Development & Engineering Centre
Benet Laboratories, Bld 115
Watervliet
NY 12189-4050, U.S.A.

Prof G. Verchery
Ecole des Mines de Saint-Etienne
158 cours Fauriel
42023 Saint-Etienne Cedex 2
France

Mr M. Wisnom
Dept of Aerospace Engineering
University of Bristol
Queens Building, University Walk
Bristol BS8 1TR

Ms J. Zugg
Ove Arup &Partners
Consulting Engineers
13 Fitzroy Street
London W1P 6BQ

DELEGATES

Mr. G. Allen,
Short Bros. plc,
Airport Road,
Belfast, BT3 9DZ,
N. Ireland.

Mr P. Bill
PDA Engineering Int.
Rowan House
Woodlands Business Village
Coronation Road
Basingstoke RG21 2JX

Mr. J. W. H. Blake,
Ex Boeing Canada,
c/o 58 Walton Road,
Harrow,
Middx, HA1 4UU.

Dr. S. E. Clift,
Dept. of Mechanical Engineering,
University of Bath,
Claverton Down,
Bath, BA2 7AY.

Mr. E. Dopper,
DSM Research,
P.O. Box 18,
6160 MD Geleen,
The Netherlands.

Mr. J. M. Giner,
AITEX (Textile Research Assoc),
Central Plaza Emilio Sala 1,
03800 Alcoy,
Spain.

Mr. F. F. Grau,
AITEX (Textile Research Assoc),
Central Plaza Emilio Sala 1,
03800 Alcoy,
Spain.

Mr. E. Greenhalgh,
Royal Aerospace Establishment,
Composite Structures,
X34 Building,
DRA Aerospace Div,
Farnborough,
Hants, GU14 6TD.

Dr. S. Grove,
ACMC,
Polytechnic South West,
Drake Circus,
Plymouth, PL4 8AA.

Mr. R. L. Hallett,
British Aerospace plc,
Civil Division,
C2 N.T.C.,
Filton House,
Filton,
Bristol, BS99 7AR.

Mr. Y. Harari,
S.P. Technologies,
Eastgate House,
Town Quay,
Southampton, SO2 7RF.

Mr. G. L. Hawkins,
Dept. of Ship Science,
University of Southampton,
Highfield,
Southampton, SO9 5NH.

Mr. A. Hughes,
Dowty Aerospace Gloucester,
Cheltenham Road,
Gloucester, GL2 9QH.

Dr. D. J. Irving,
FEA,
Forge House,
66 High St,
Kingston, Surrey KT1 1HN

Mr. S. Lahiff,
Royal Aerospace Establishment,
Composite Structures,
X34 Building,
DRA Aerospace Div,
Farnborough,
Hants, GU14 6TD.

Mr. L. Marks,
Desktop Engineering,
Stanton Harcourt Road,
Eynsham,
Oxford, OX8 1TW.

Mr. M. Oxley,
British Aerospace plc,
MO66 Comet Way,
Hatfield,
Herts, AL10 9TL.

IMPERIAL COLLEGE CENTRE FOR COMPOSITE MATERIALS
WORKSHOP: FINITE ELEMENT METHODS FOR COMPOSITES,
5TH & 6TH SEPTEMBER 1991.

Mr. D. R. Payne,
Reynard Technology,
Telford Road,
Bicester,
Oxon, OX6 0UY.

Mr. T. R. Pegler,
British Aerospace plc,
Civil Division,
A3 N.T.C.,
Filton House,
Filton, Bristol BS99 7AR

Dr. S. Shamasundar,
Chemical Engineering Dept,
Imperial College.

Dr. C. Smith,
Courtaulds Research,
101 Lockhurst Lane,
Coventry, CV6 5RS.

Mr. M. Smith,
W. S. Atkins Engineering Sciences Ltd.,
Woodcote Grove,
Ashley Road,
Epsom,
Surrey, KT18 5BW.

Dr. D. L. G. Sturgeon,
E. I. Du Pont de Nemours,
Chestnut Run/Bldg. 702,
P.O. Box 80/02,
Wilmington,
Delaware 19805,
USA.

Mr. A. Vince,
Westland Helicopters Ltd.,
(Box 234),
Yeovil,
Somerset, BA20 2YB.

Mr. A. L. Webb,
Maragate Ltd.,
Steppingley New Road,
Chilworth,
Guildford,
Surrey, GU4 8LZ.

Mr. R. A. S. Wright,
Aerotex, CSIR,
P.O. Box 395,
Pretoria,
South Africa.

Mr N. Zahlan
ICI Advanced Materials
Wilton Materials Research Centre
P O Box 90, Wilton
Middlesbrough,
Cleveland TS6 8JE

OTHER IC STAFF AND STUDENTS

Prof. G.A.O. Davies,
Aeronautics Dept.

Prof. M. Crisfield,
Aeronautics Dept.

Dr. P. Robinson,
Aeronautics Dept.

Mr. S.J. Kukula,
Aeronautics Dept.

Mr. G. Giaroli,
Civil Engineering Dept.

Mr. E.W. Godwin,
Composites Centre.

Mr. A.K. Molyneaux,
Composites Centre.

SOFTWARE SUPPLIERS

STRUCOM
Mr D. Ellis & Mr S. Morrison
Structures and Computers Ltd
Strucom House
188-186 Canterbury Road
Croydon CR0 3HF, Surrey

FEA
Mr G. Duffett
Forge House
66 High Street
Kingston upon Thames
Surrey KT1 1HN

DESKTOP ENGINEERING
Mr R. Greensmith
Stanton Harcourt Road
Eynsham
Oxford OX8 1TW