NPL Report MATC(A)96

April 2002

An Internet Accessible System for Simulating Damage in Composite Laminates During Bending Deformation

N J McCormick, M Gower and L N McCartney
Materials Centre
National Physical Laboratory
Teddington, Middlesex TW11 0LW, UK

ABSTRACT

This report describes the application of NPL-developed stress transfer models, of damage in laminated cross-ply composites subject to bending, to a web-based simulation system that is designed to offer the user an easy method of using the models, that can assist in the design of damage resistant composite components. The simulation allows the user to construct a virtual laminate, to visualize its appearance, and to observe damage development during loading that is governed by energy principles. In addition, quantitative design data are accessible as graphical output and in digital file-based form. The software system integrates FORTRAN software with a graphical web-based interface that includes the use of 3D visualisations based on virtual reality software routines. The simulation can be found at:

http://materials.npl.co.uk/bend/energy.html

ISSN 1473 2734

National Physical Laboratory
Teddington, Middlesex TW11 0LW, UK

Approved on behalf of Managing Director, NPL,
by Dr C Lea, Head, NPL Materials Centre

CONTENTS

INTRODUCTION

This report describes the NPL Internet Simulation system that enables the energy-based prediction of damage in composite laminates under bending loads, and the effect of damage on laminate thermo-elastic properties. The system allows a user to build up a model of a laminate from a series of pre-built plies and then apply a bending moment to the laminate. The system comprises an interfacing software module and a calculation engine that runs on an NPL server.

An Internet browser window on the user's machine is used for the user interface, with an area for parameter input and a three dimensional scene viewer to display the behaviour of the laminate. Once the model parameters are chosen a simulation can be undertaken. This simulation runs on an NPL server, thus ensuring its quality and, on completion, the data is sent back to the user's machine. The user can then replay the simulation, in the three-dimensional scene window, by increasing the applied bending moment using a screen slider that is operated by a mouse. The loading results in ply cracking in one or more plies as the laminate is progressively deformed. An important feature of the model is that is predicts anticlastic bending, and this is accurately displayed in the three dimensional scene. After completing the calculation, a separate window can be opened that will display graphs predicting the macroscopic behavior of the laminate.

The simulation can be found at

http://materials.npl.co.uk/bend/energy.html

At the start of the project, it was anticipated that a version of this software would be developed that could be loaded from a CD and run on users' machines locally. Although this is an attractive idea, there were several reasons why this approach was not pursued. The main reason is concerned with the difficulty of porting the software to users, as it would have to run on many different personal computers having different operating systems and hardware configurations. For the system to run effectively on different platforms, it would need to have a calculation engine, user interface and a three dimensional scene viewer that could run on many potentially different computers. This could not be achieved within the resources of the project and would require the use of standardized interfaces and modules. Therefore, it was decided that the software system should be run using a web browser and the standardized Java and VRML languages. These components can run on practically all currently available computing platforms, and are being continuously developed by third parties and ported to any new operating systems as they become available. This removes a tremendous burden of software development work and enabled the significant progress that is highlighted in this document.

One disadvantage of this approach is that it is nearly impossible for users to interact with software executables held locally on a remote machine using their web browser. This is a deliberate security policy that has been implemented to stop malicious damage to the client computer.

A benefit of this policy is that the users of the system can interact with a well validated version of the software, running on an NPL server, that has been extensively tested and will provide reliable results, rather than with an untried and tested combination of hardware and software running on their own machines.

2 MATHEMATICAL MODEL

The basis of the mathematical model, on which the simulation methodology has been developed, is a stress transfer analysis that can predict accurately the stress and displacement distributions in a damaged multi-layered cross-ply laminate that is subject to biaxial in-plane and out-of-plane loading. The damage is assumed to be in the form of a regular array of ply cracks in one or more of the 90° plies of the laminate. The methodology takes full account of thermal residual stresses that arise because of thermal expansion mismatch effects between adjacent plies having different orientations. Appendix A provides some model validation data developed during the project, illustrating that the stress transfer model can predict results that are in close agreement with those obtained using the boundary element technique. The mathematical details of the model have been described in reference [1]. The stress analysis is used to predict ply crack formation using energy principles. The mathematical analysis for such predictions has been described in reference [2]. Two FORTRAN 77 computer programs have been written. The first enables the calculation of the stress and displacement fields at all points in the damaged laminate. The second program performs the energy balance analysis needed to predict the conditions for which progressive ply cracking can occur.

3 INTERNET INTERFACE

The software system is divided into two parts to maximize the interactivity that is available to users over the Internet. The first part of the system combines an interface, written in the C computer language, that enables easy communication with the main simulation that comprises a suite of FORTRAN77 executables. This interface runs as a server process and will respond to any incoming messages from a user and act accordingly, returning information as required. The system has been designed to work in this manner so as to minimize the amount of information that needs to be transferred over the Internet, as this is the biggest bottleneck to performance, and to maximize the interactivity of the system. Additionally the main simulation software, which is written in Fortran, is essentially the same software as the well-validated software that is used at NPL to develop and validate the mathematical models. Consequently, the simulation software can easily be updated as new features become available. Any numerical problems, when running this software on different platforms/ machines, can also be minimized when using this approach.

The user has the simulation interface running on their machine in an Internet browser window. Using this browser to run this part of the system has the advantage of benefiting from a well developed piece of software that is familiar to most users, and is relatively standardized, well behaved and can run on many different platforms. However, there have been problems in the past over the non-conformance to accepted standards of the different browsers that are available. This is being largely overcome by the introduction of the latest generation of browsers that should soon bring more benefits to the user, and aid the developer with the closer adherence to the accepted standards for browsers.

The form of the interface chosen for this system is a development of the interface used in previous composite laminate damage applications, and it has been streamlined to improve robustness and reliability. The important elements are still present and these are; a three dimensional scene that the user can manipulate containing a representation of the laminate, and a user interface that uses mouse-controlled sliders and buttons to operate the system. In this system additional windows are spawned as required to enable the input of additional parameters that are required by the system, and to present to the user graphical results generated by the system. A new visual feature of the system is its ability to portray changes of curvature of the laminate during loading and progressive damage growth.

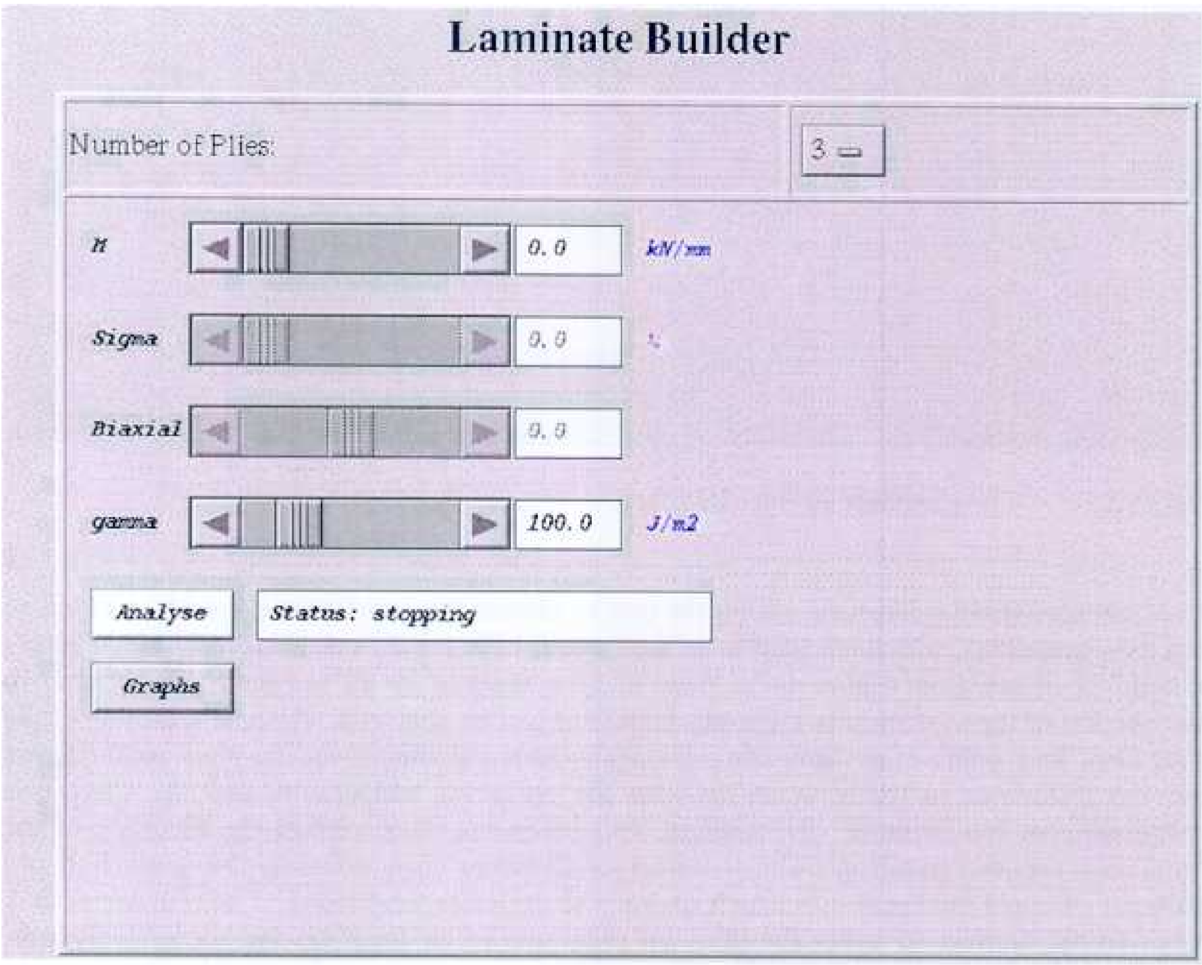


Figure 1. The input parameter interface for the simulation.

Figure 1. shows the interface that is used to input simulation parameters. The first parameter that is required is the number of plies that make up the laminate. Once this parameter is chosen an additional window appears containing the Laminate Builder Interface (see Figure 2). The second and third parameter input boxes are disabled and greyed out currently as they will be needed for future developments of the system involving the control of combined bending and tensile loading, and transverse loading. The fourth parameter that must be entered, labelled 'gamma' is the effective fracture energy for ply cracking.

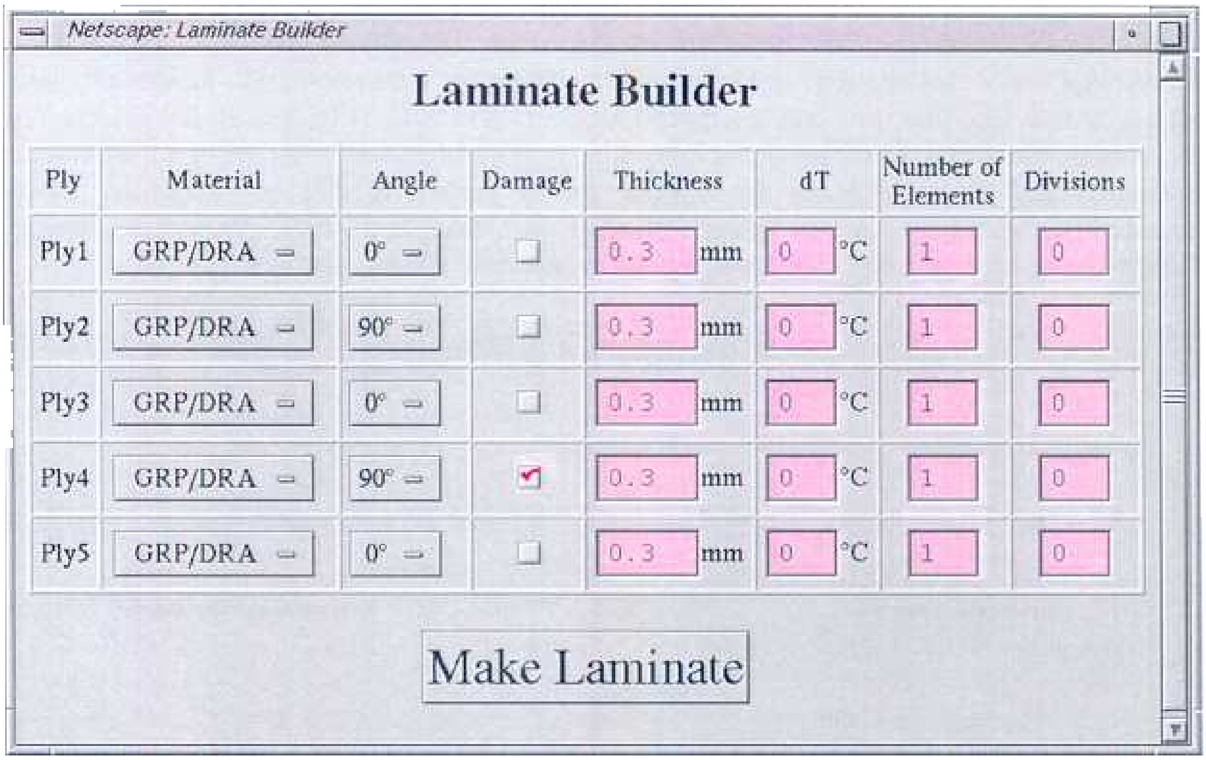


Figure 2. The Laminate Builder Interface

The Laminate Builder interface allows the user to choose the constituent plies of the laminate and their properties, which are used to create a control file for the calculation engine. The ply material is chosen from a drop down menu and the angle (0^o or 90^o) of the ply is chosen. In this version of the system only cross-ply laminates can be analysed. Also, at least one of the plies must be capable of accumulating damage. Other parameters for the plies are thickness and the difference in temperature between the operation temperature and the stress-free temperature of the laminate. In addition, two parameters that control the accuracy of the simulation are also required. The 'Number of Elements' box refers to the number of ply elements of equal thickness into which each ply specified is subdivided. The 'Divisions' box refers to the number of times the plies on each side of an interface are successively sub-divided. These two parameters define a ply refinement procedure that enables the through-thickness variation of the stress field to be adequately modelled, and consequently enables highly accurate solutions (see Appendix A for validation data where the model is compared to solutions obtained using the boundary element technique). Allowing the difference in temperature to vary for each ply means that the temperature gradients, leading to bending, can be modelled using the system, although this has not been investigated in detail.

Of the other parameters in the main interface only 'gamma' is required to start the simulation. This is the fracture energy for ply crack growth that is applied to each potential crack site. In the current version of the software this energy has a unique value for each potential crack formation site. A consequence of this is that when the damage first occurs it is in the form of an equally spaced array of ply cracks having a specific density that is a function of the thickness and properties of the plies in the laminate. Damage can only increase for this idealized case by the formation of new ply cracks at the mid-points between existing ply cracks, a process that occurs successively as the loading is increased. Introducing statistical variability of the fracture energy will lead to more realistic damage growth patterns where ply cracks form one at a time.

The 'Analyse' button starts the simulation and the 'Status' box reports on progress with the calculations. The 'Graphs' button spawns additional windows that can contain graphs of various laminate properties such as the crack density versus moment, etc.

4 BEND APPLICATION

An example is now given of a simulation that was undertaken using the website

http://materials.npl.co.uk/bend/energy.html

In practice, operation of the simulation is simple and intuitive. The first step for the user is to define the laminate in terms of pre-built plies and their properties. In the example below a ply has been generated using five plies with damage confined to the fourth ply from the bottom, which is regarded as being in the tensile region during bending.

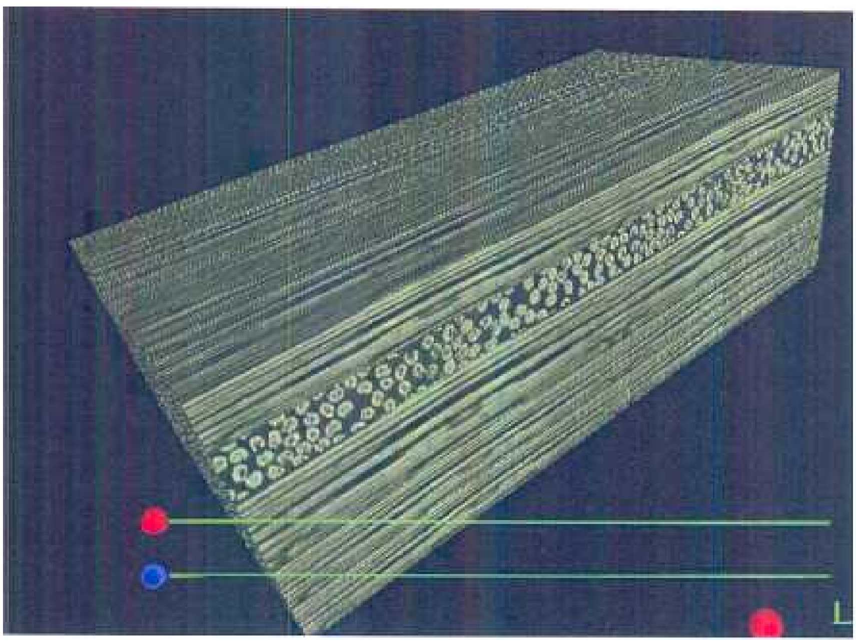


Figure 3. 3D image of a laminate that is ready for 'Virtual Testing'

The next step is to choose for the simulation a value for 'Gamma', (i.e. the effective fracture energy for ply cracking). Once this has been done the job can be submitted. After a short wait for the results, typically less than 30 seconds, the system can be used to play back the simulation that has just been completed.

As the parameter 'M' is changed using the slider, the laminate will start to bend. At critical values of the applied bending moment, the ply that is allowed to crack will suddenly form a regular array of ply cracks. Further loading leads to a successive doubling of the ply crack density at a discrete set of applied bending moments. This process continues until a maximum radius of curvature allowed by the simulation is reached, as shown in Figure 4.

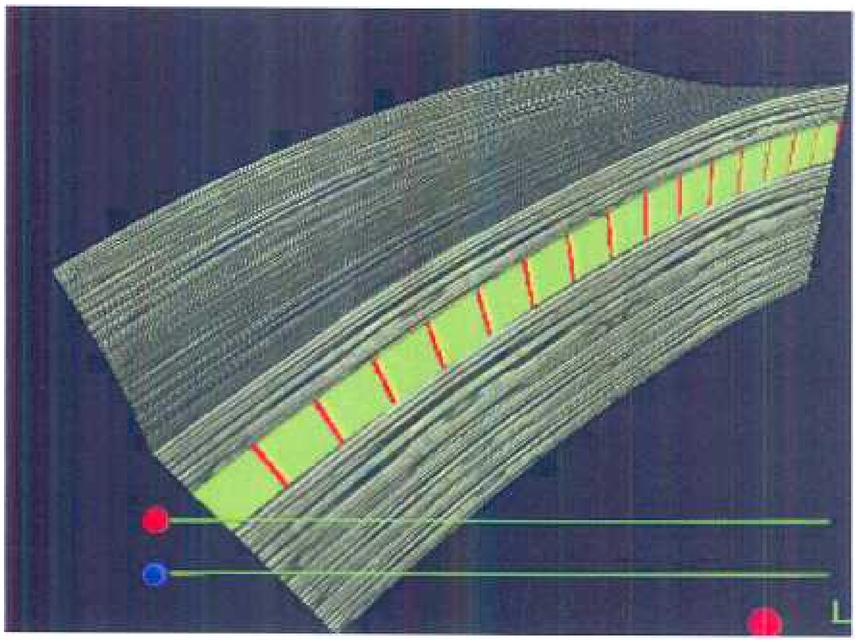


Figure 4. A damaged laminate showing anticlastic bending.

A variety of graphs can be generated describing quantitatively the macroscopic behaviour of the laminate during loading. Examples are given in Figures 5-8. In the graphs Rho is the crack density per mm.

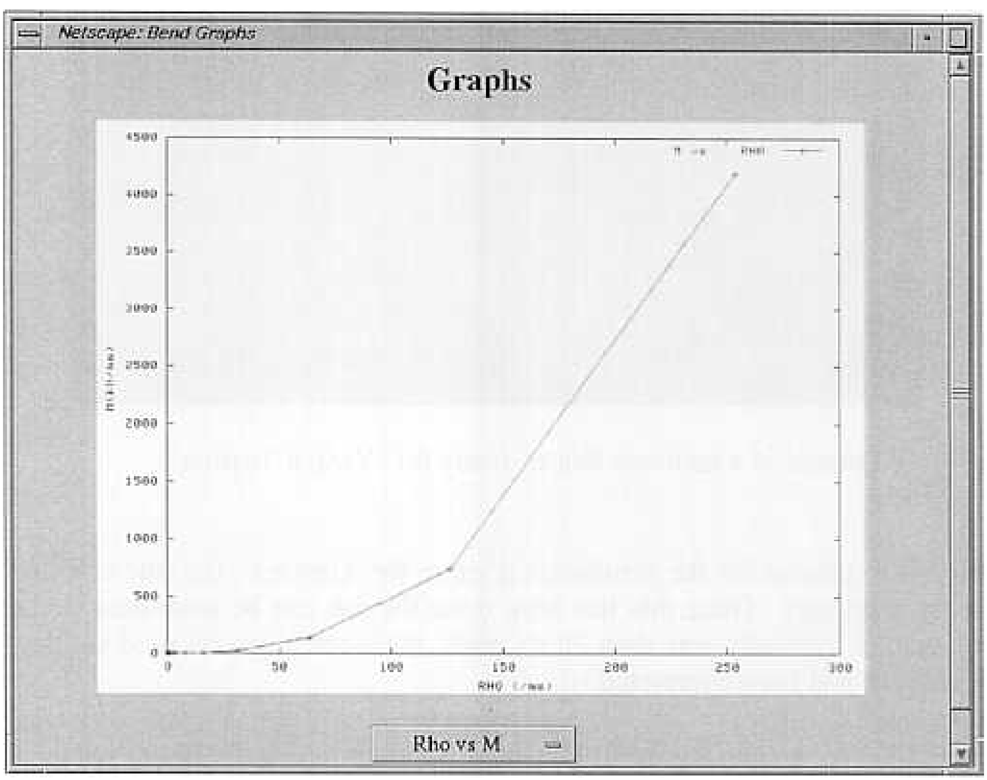
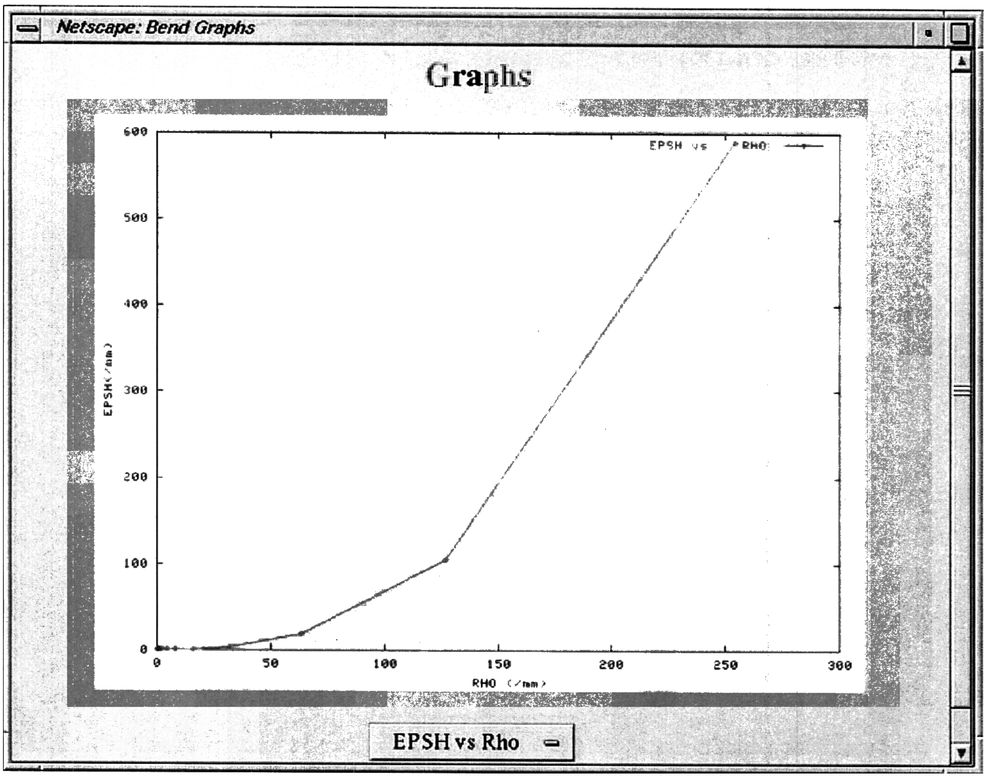
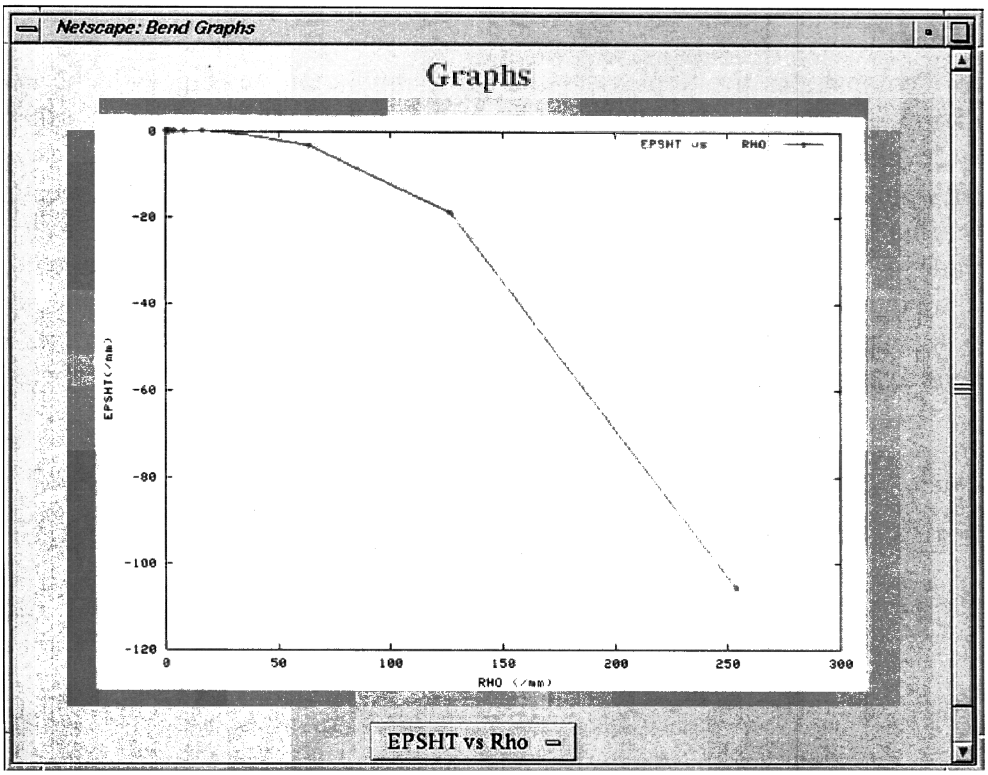


Figure 5. Moment per unit cross-sectional area versus ply crack density.



Re Cu ity



mi acl ity

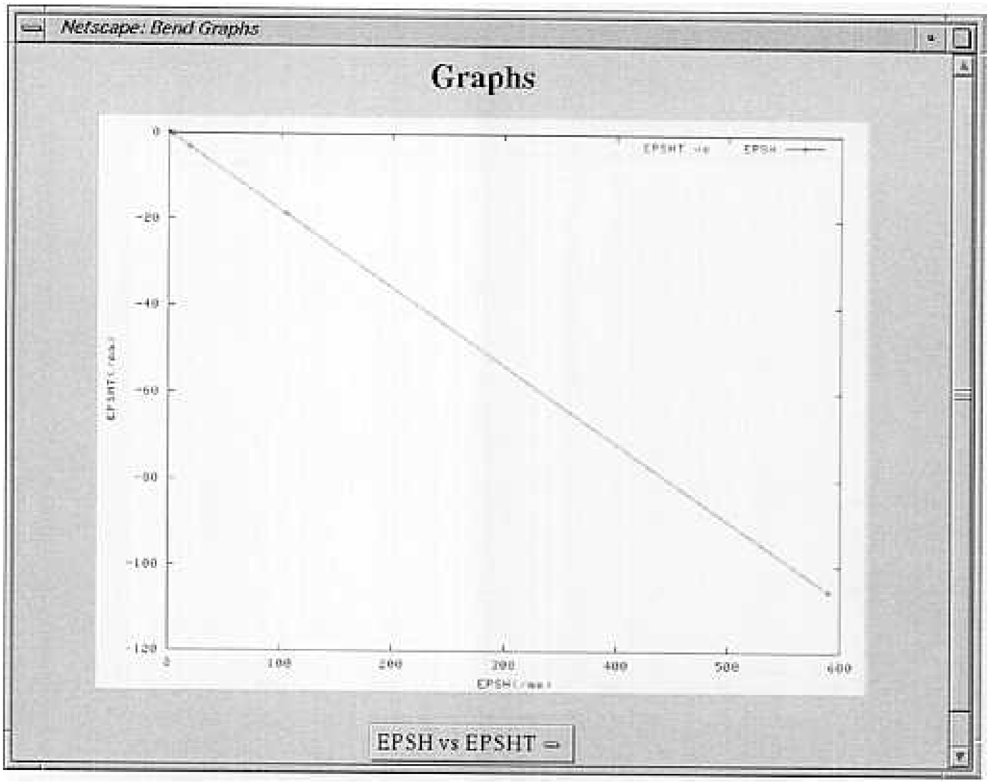


Figure 8. Transverse bending strain versus axial bending strain for the damaged laminate.

5 CONCLUSIONS AND FUTURE WORK

This system demonstrates the implementation of a simulation dealing with the bending of composite laminates and damage evolution in the laminate during loading. Further work is required to develop the model to the point where general symmetric multidirectional plies, (i.e. not just limited to 0°/90° orientations), can be successfully modelled. A stochastic factor, as used in earlier laminate simulations to model damage evolution at a smaller scale will also need to be added. The current interface and three dimensional scene generation methodology can easily cope with the additional requirements of this development of the modelling, however, a high performance computer will be required to display individual cracks that form in the plies since they will be of quite complex shape.

REFERENCES

McCartney L N, 'Stress transfer mechanics for multiple ply cross-ply laminates subject to bending', summary in Proceedings of 6th International Conference on the Deformation and Fracture of Composites, Manchester, April 2001, pp. 57-66.

2. McCartney L N & Byrne M J W, 'Energy balance method for predicting cracking in cross-ply laminates during bend deformation', Proc. 10th Int. Conf. on Fracture (ICF-10), *Advances in Fracture Research*, Honolulu, 2-6 Dec. 2001.

APPENDIX A

Validation data for model for cross-ply laminates subject to bending

The stress transfer model developed at NPL that has been used to predict the effects of ply cracking on the thermoelastic constants of damaged laminates subject to bend deformation has been validated by comparison of predicted stress distributions with results obtained using the commercial boundary element system BEASY. A boundary element method has been used in the comparison as it can handle the singularities that must occur at the ply crack tips for linear elastic solutions.

The thickness of the plies in a $[0/90/0/90/0]_s$ GRP laminate has been taken as 0.2 mm. The ply properties assumed for the comparison are those of a typical GRP and the values are given by:

$E_A = 45.6$ GPa	$E_T = 16.2$ GPa	$\nu_A = 0.278$	$\nu_T = 0.4$
$\mu_A = 5.83$ GPa	$\alpha_A = 8.6 \times 10^{-6}/^{o}C$	$\alpha_T = 26.4 \times 10^{-6}/^{o}C$	

Where E is the elastic modulus, ν is the Poisson ratio, μ is the shear modulus and α the thermal expansion coefficient. The subscripts A and T apply to the axial and transverse directions.

In Figure A1 is shown the interfacial shear stress distribution where y = 0 is at the mid-plane between two neighbouring ply cracks that are separated by a distance of 1 mm, and where one ply crack is located in the 90^o ply at y = 0.5 mm. It is seen that the NPL model prediction agrees very well with the BEASY result.

Figure A2 shows the interfacial axial stress distributions in the 90^o and 0^o plies, respectively again indicating very good agreement between the NPL model and the BEASY results.

It is concluded from the results shown in Figs. A1 and A2 that the stress-transfer model used in the web-based simulation system is accurate, and can be used with confidence to predict progressive ply cracking governed by energy balance principles.

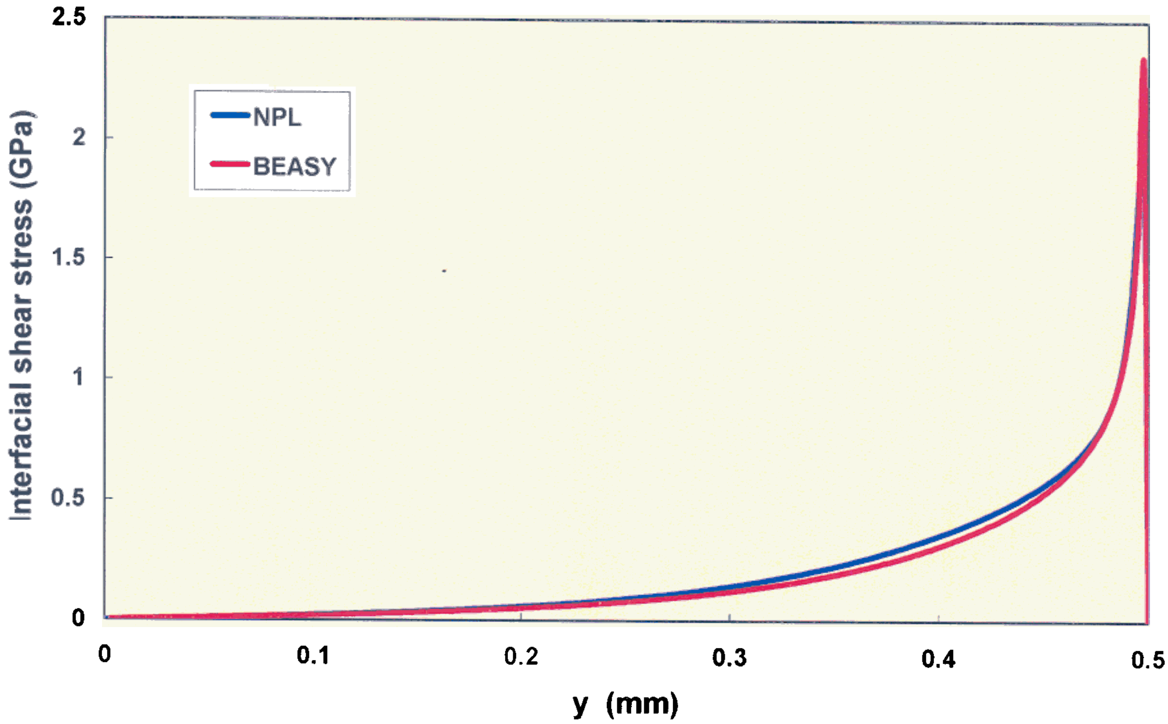


Figure A1 Comparison of predictions for the interfacial shear stress distribution.

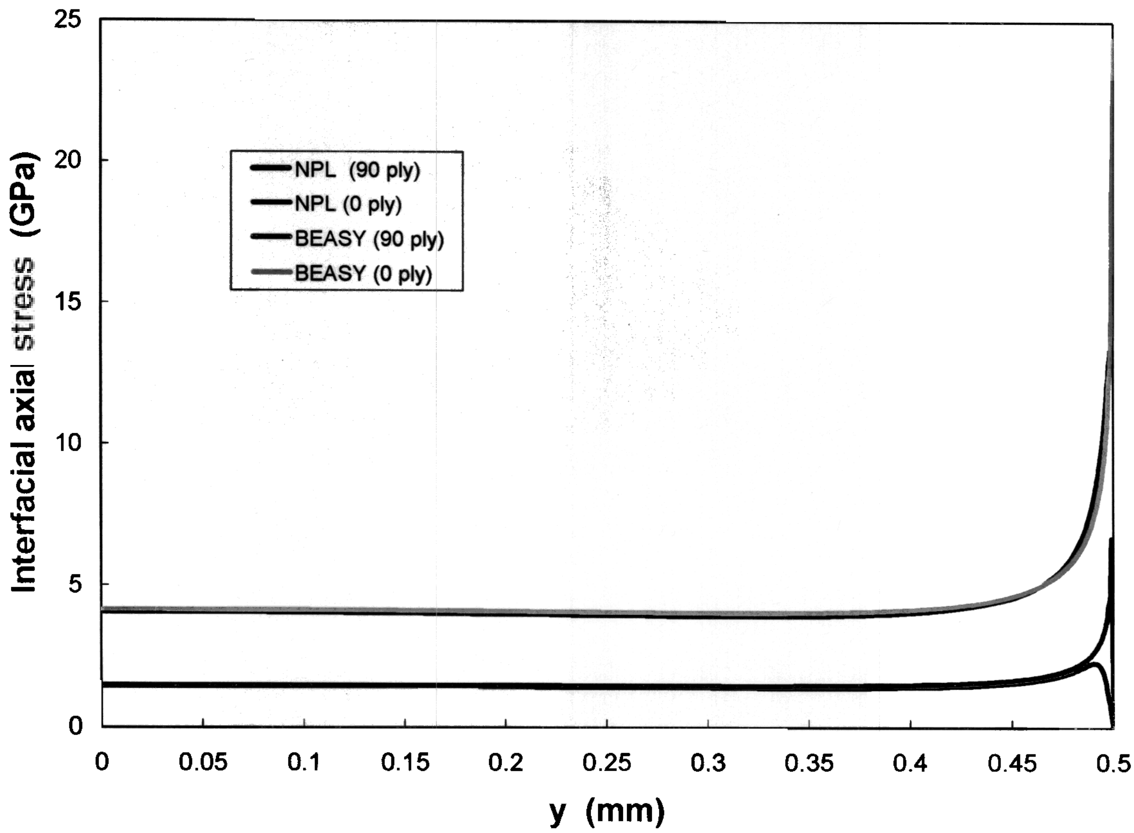


Figure A2 Predictions for interfacial axial stress distribution in the 0° and 90° plies.