MICROMECHANICAL FINITE ELEMENT STUDY OF IMPACT DAMAGED FRACTURED FIBRES UNDER COMPRESSION

R. Craven, R. Olsson and B. G. Falzon

Imperial College London
South Kensington, London, SW7 2AZ, England.
r.craven06@imperial.ac.uk

ABSTRACT

This paper examines the average stress required for a fractured fibre to penetrate the matrix between other fibres, to determine the in-plane compressive stiffness of an impact damaged plate with fractured fibres. The paper also uses basic prediction of kink band initiation stress to determine whether fibre penetration or kink banding would be more likely to occur in impact damaged laminates in compression, in the region of the fibre fractures. Initial studies using 2D and 3D models have been conducted suggesting that fibre penetration occurs at a stress 10 times lower than fibre kinking would occur, but matrix material properties, fibre spacing, fibre waviness are likely to have large influence over the penetration stress, so fibre kinking could still occur under some conditions. The Drucker-Prager plasticity model used in the analysis was initially explored in a parametric study of plastic deformation of various polymer resins under tri-axial stress.

1. INTRODUCTION

Impact damage is often considered one of the fundamental problems with composite structures, as it degrades the normally excellent in-plane properties of composite laminates. Composites are very susceptible to damage because of their sensitivity to load in the out-of-plane direction, [1]. The damage caused by impact at low velocities is particularly hard to detect because it leaves no visual surface damage while causing severe internal damage which can cause the strength of the structure to drop to less than half that of an undamaged structure.

In this study the effect of the dominating in-plane feature of impact damage, namely fibre fracture, is being considered in compression of a ply. This follows previous work conducted on in-plane damage mechanisms in tension, [2]. The aim of this work is to gain an understanding of the behaviour of fractured fibres under compression and their effect on the local stress field in the impact damage region. This is a part of a larger project to define a homogenised nonlinear model of the mechanical behaviour of impact damaged laminates that can be used to represent impact damage in large structural Finite Element (FE) models.

In undamaged laminates compressive failure is caused by microbuckling and kink banding. This topic has been widely studied over the past forty years, both experimentally and analytically, [3]–[4] and more recently using FE models, [5]. In impact damaged laminates, delamination and local/global buckling are considered to be the main failure modes, again this has been modelled extensively using analytical, experimental and finite element methods, [6]–[8]. These studies have looked at replicating impact damage either artificially in experiments or using FE models, and have replaced the impact damage with single or multiple circular delaminations. While these models predict the global buckling load within a 10-15% range of compared with a real impact damage, there is a marked difference between the predicted and observed local buckling behaviour of the impact damage, Fig. 1.

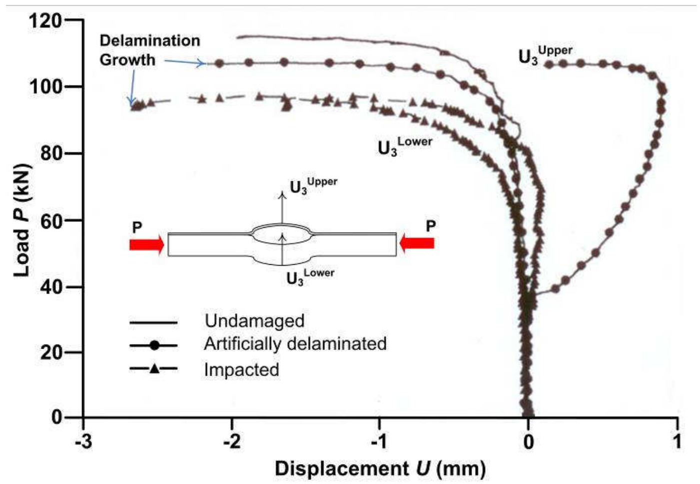

Figure1: Load vs. out-of-plane displacement from experiments, [6]

This was thought to be due to the circular representation of the delamination rather than the peanut shapes that actually occur in real impact. Suemasu [9] attempted more realistic shapes with wedge shaped delaminations without success. If delamination shape was the only reason for the discrepancy between model and reality the bifurcation point, i.e. where the delamination buckling starts, should be much higher for the real impact damage as the width of each of the delaminated regions is much smaller. Zeng and Olsson [10] approached the problem differently and examined the effect of a single circular delamination model with a soft inclusion region representing the damage caused by fibre fracture and matrix cracking. The effect on the buckling load was, however very limited and did not agree with the experimental observations in Fig.1. This suggests that both the shape, number of delaminations and the in-plane damage are important to correctly model the compressive failure. The current work is the first stage of attempting to accurately model impact damage in compression using Finite Element Analysis (FEA).

2. FRACTURED FIBRE MODEL

The aim of these models is to gain an understanding of the stress/strain distribution within regions of fractured fibres. Then the reduction in compressive stiffness caused by these fractures can be quantified and used with a larger homogenised model of impact damage and delamination. To determine this information, modelling of individual fibres embedded in matrix material is required. Typical fibre fracture cracks would encompass hundreds of broken fibres, which is obviously not realistic to model due to computational constraints. A smaller number of fibres are modelled to represent a unit cell of the larger fibre fracture crack. Simplifications include modelling the fibres as perfect cylinders even though in reality they are irregular in cross section. The initial fibre spacing is also assumed regular for simplicity of modelling.

The boundary conditions applied have been chosen so that the model replicates a unit cell, surrounded by other broken fibres. The displacement is applied along the axis of the fibres, the fibres are prevented from moving sideways (in-plane) but are be allowed to move out-of-pane, i.e. in the thickness direction of the laminate. The fibres themselves are modelled as linear elastic and assumed not to fail, while the matrix material undergoes large plastic deformation, under a triaxial stress state for which the Drucker-Prager plasticity model is used, [11].

To model this phenomenon, and to investigate parameters such as fibre diameter, fibre spacing and Drucker-Prager parameters two models have been produced: A Two Dimensional (2D) model, Fig. 2a, for parametric studies and a Three dimensional (3D) unit cell model, Fig. 2b, to capture the full stress strain distribution of one fibre pushing through and deforming the matrix between two other fibres.

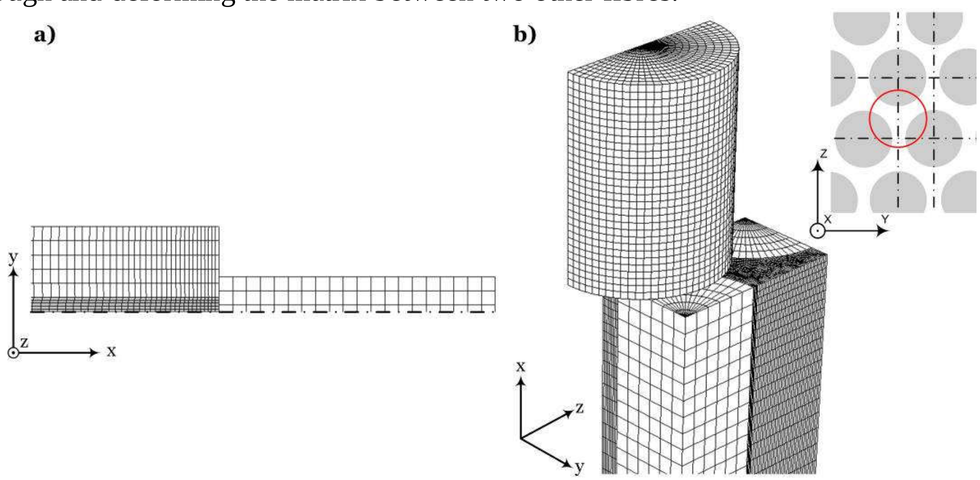

Figure 2: a) 2D 3 fibre compression model b) 3D unit cell compression model

It is also suggested that the fibres could kink rather than push through each other forming a chevron shape through the thickness of the laminate, Fig. 3.

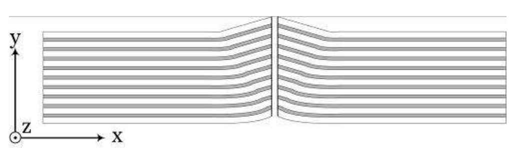

Figure 3: Fibre chevron kinking an alternative failure mode

Rosen [4] derived a formula for compressive strength of composites Equation 1 which when divided by an empirical correction factor of 4, is considered good approximation of the kinking stress, [12].

$$\sigma_k = \left(G_m / 1 - v_f\right) \quad (1)$$

This would give an approximate kinking stress of σ_k = 900 MPa, which provides the stress at which the fibres supported by an elastic material would buckle, giving an upper limit to the stress required for the chevron kinking to occur. A 2D FE model has been developed based upon previous kink band models. The model allows kink banding and/or fibre push-through to occur, and also investigation of the effects of parameters such as fibre spacing, material properties, fibre eccentricity and fibre waviness on the failure mode.

3. DRUCKER-PRAGER MATRIX MODEL

The stress state within the matrix between the fibres is highly complex due to its triaxial nature. The matrix also undergoes large deformations when the fibres push-through, causing plastic deformation of the matrix. This means a simple yield criterion such as von Mises' would not accurately capture the matrix behaviour so a more complex yield criterion such as Drucker Prager is required. This model has been successfully used to model plastic deformation, [11] and has been used in kink band models, [5]. The Drucker Prager yielding model is fully implemented within ABAQUS and requires three parameters: The angle of friction, β which is a measure of the sensitivity of the material yield stress to hydrostatic pressure; the dilation angle, ψ which measures the compressibility of yielded polymer, and the flow stress ratio, κ which is the ratio of compressive/tensile yield strengths.

The parameters β and κ are determined from the yield stresses; in tension σ_T, shear σ_S and compression σ_C. The flow stress ratio κ is the ratio of yield stresses giving a value between 0.778 and 1.0 where one represents von Mises' failure surface, i.e. the yield stress is the same for tension and compression. ψ is found from the Poisson's ratio ν_P under plastic conditions as shown by Equation 4.

$$\beta = \tan^{-1}\left(3\left[\left(\sqrt{3}\,\sigma_S/\sigma_T\right)-1\right]\right) \text{ using tensile and shear test data} \tag{2a}$$

$$\beta = \tan^{-1}\left(3\left[\left(\sigma_C/\sigma_T\right)-1/\left(\sigma_C/\sigma_T\right)+1\right]\right) \text{ using tensile and comp test data} \tag{2b}$$

$$\beta = \tan^{-1}\left(1-\left[\sqrt{3}\,\sigma_S/\sigma_C\right]\right) \text{ using shear and comp test data} \tag{2c}$$

$$\kappa = \left|\sigma_C/\sigma_T\right| \tag{3}$$

$$\psi = \tan^{-1}\left(3\left(1-2\nu_P\right)/2\left(1+\nu_P\right)\right) \tag{4}$$

Only a limited amount of data required for calculating these parameters for epoxy resins was found in the literature. For this reason a parametric study was conducted to understand the relationship between β, ψ and κ and the yield stress and plastic Poisson's ratio. ψ and κ showed linear or quasi-linear relationships with their material property inputs. β, however, showed a nonlinear behaviour and also proved to be dependent on the equation used. The idea behind Equations 2a, b, c is that β can be calculated using any two of the yield stresses, but in reality the choice of equation affects the value of β, see Fig.4

An FE model of simple cubes loaded in compression and shear under plane stress and plane strain was conducted to determine the effect of the different values of β, ψ and κ on the material behaviour. The general trend in plane strain was for the compressive stress to be higher in the compressive case than in the shear case and for the compressive loading to be more sensitive to change in parameters particularly in the plane strain condition. ψ was the least significant of the parameters and β the most significant, in terms of changing the stress strain behaviour of the cube.

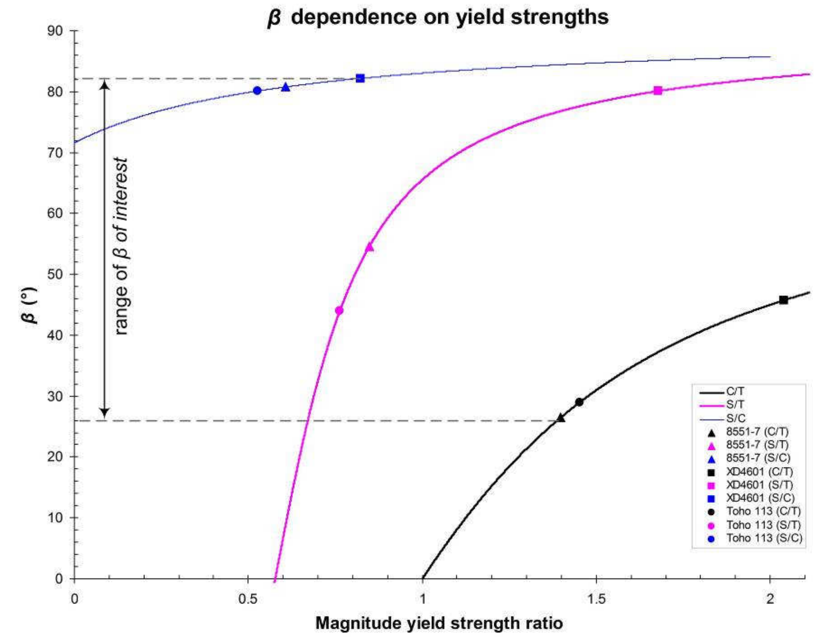

Figure 4 Dependence of β on yield strengths and example values of β for different epoxies.

It should also be noted that the model failed to converge for values of β outside of the range 15º-65º which would make some of the calculated values of β unusable, so this needs to be investigated further on a fibre matrix model to ensure it is not a geometric problem. A representative plot of the general trends is shown in Fig. 5, where plane stress PS and plane strain PE cases are shown.

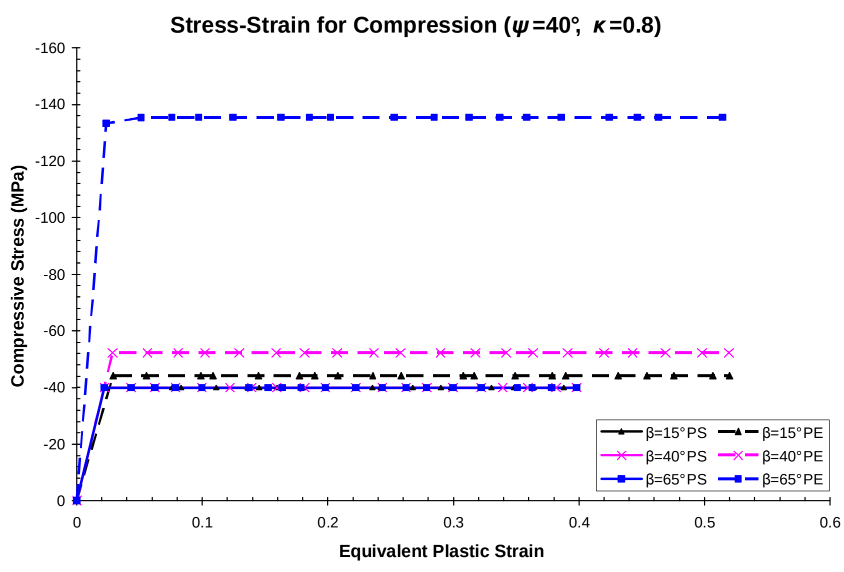

Figure 5 Compressive stress strain plot for varying values of β plane stress and plane strain cases.

4. FINITE ELEMENT MODELS

The FE models were constructed in the ABAQUS 6.6 CAE module and solved using the ABAQUS 6.6 Explicit solver. The fibres are linear elastic and assumed not to fail. The matrix is assumed to be elastic/plastic and the plasticity is modelled using the Drucker Prager material model. The material properties for IM7 fibres and 8551-7 matrix have been taken from [13] and the parameters for the Drucker Prager model calculated as described in section 3 with β determined from Eqn. 2a.

The 2D model boundary conditions are encastre at the far end of the fibre and the penetrating fibre is allowed to move in the x-direction only. Due to symmetry only one wedged fibre and half of the penetrating fibre are modelled. A velocity is initially applied in y-direction to separate the wedging fibres before the penetrating fibre pushes through into the matrix in the x-direction again under applied velocity, see Fig. 2a.

The 3D model is based on a unit cell of fibres and matrix, wherever possible symmetrical boundary conditions are applied. Encastre (clamped) boundary conditions are again used at the opposite end of the unit away from the penetration. Velocities in the z-direction are initially applied to the top and bottom edges of the unit cell model in the same way they are in the 2D model separating the fibres to allow the third fibre to penetrate the matrix, Fig. 2b. Note that motion in the y-direction was prevented on the boundaries to simulate the conditions in the interior of a laminate, while motion in the laminate thickness direction (z) was permitted, but not enforced except during the initial separation phase.

The default 3D stress elements C3D8R for ABAQUS Explicit are used and similarly in 2D CPE4R are used. The fibres are attached to the matrix material using tie constraints, which is partly to simplify the model but also because it has been observed from published work that with modern toughened resins cracking along the fibre matrix interphase does not appear to occur.

4.1 Two dimensional single fibre model

The 2D model is optimised to run on a Pentium 4 with 1GB of RAM. The mesh is refined to the maximum capability of the computer (by reducing the number of dof) which results in a solution time of 30 – 40 minutes for this model. Consequently, the model has been used for a parametric study initially looking at the effects of the different equations for calculating β for the Drucker Prager model, which was found to cause a minimal difference on the stress strain plot to within 5% using Eqns. 2a or 2b.This suggests there is a limited effect but the equation used should depend on available data and the loading conditions of the material in the model. However, as with the Drucker Prager cube models the values of β calculated using Eqn. 2c, the model fails to converge suggesting that this equation is invalid.

The other parametric study conducted looked at difference in matrix thickness in the 2D model, which represents change in local volume fraction of the matrix. This could be due to increased fibre volume fraction in the ply, or the variation in fibre spacing within a ply of constant fibre volume fraction, see Fig. 6.

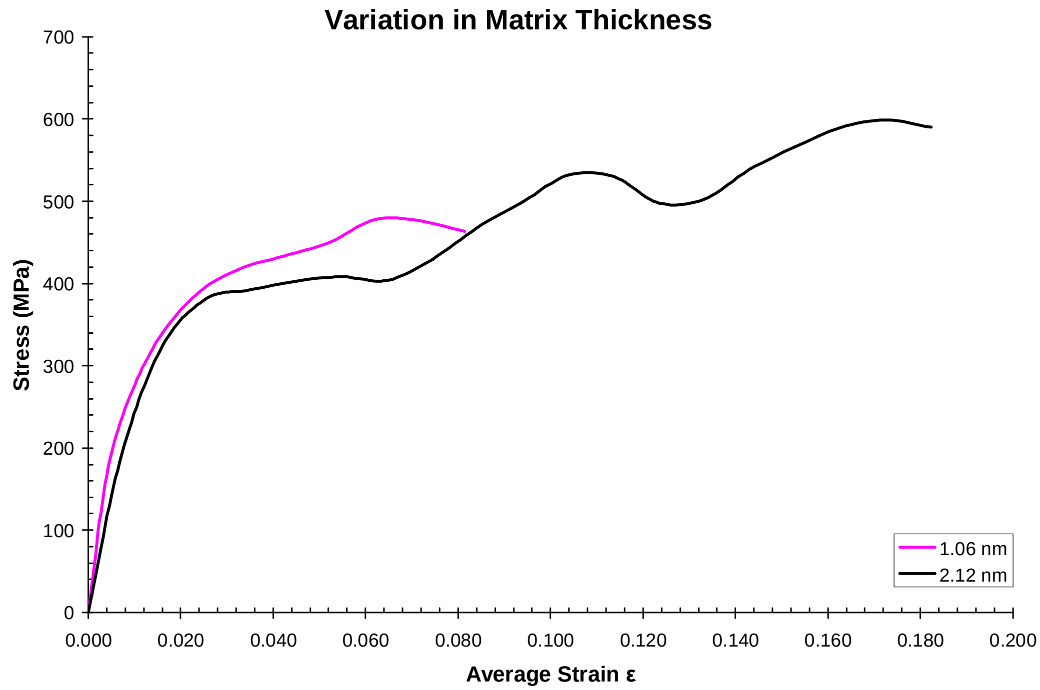

Figure 6: Change in matrix thickness

This shows that the stress required for fibre penetration is only very slightly increased for the thinnest matrix thickness. It should also be noted that the model with the thinner matrix material failed at a much lower strain, which was due to numerical errors caused by large deformation in the elements in the matrix material. The strain is defined for all models, as the distance the penetrating fibre has penetrated divided by ten fibre diameters i.e. the approximate thickness of a ply see Eqn. 5.

$$\varepsilon_{Avg} = u_{fibre} / L \qquad (5)$$

where ε_{avg} is the average strain apparent in the model, u_{fibre} is the distance the penetrating fibre has moved into the matrix and L is the normalising length to calculate the strain which is assumed to be ten fibre diameters.

4.2 Three dimensional unit cell model

The 3D unit cell model provides a more accurate model of the behaviour of matrix deformation and fibre penetration because it can fully capture the tri-axial stress state that exists within the matrix. However even on an Intel Core 2 Duo processor with 4GB of RAM the solution time is 25 – 30 hours which makes this model infeasible for doing a large number of runs in a parametric study. The other problem with this model is that excessive deformation of the elements describing the matrix material causes numerical errors and failures in the model. This means that the results for the 3D model have not advanced as far as the results for the 2D model, and the initial results are not a realistic measure of the stress because the fibre has yet to penetrate sufficiently so that the fibre comes into contact with matrix across the entire width of the fibre. In the current initial analysis the contact area between fibre and penetrating matrix is low so that the average stress resulting from matrix yielding is severely underestimated. Fig.7, shows the plastic strains in the matrix as the fibre penetration commences. The fibres are modelled as linear elastic hence show no plastic strain. The cupping in the matrix caused by the two fibres in unit cell being split apart can be clearly seen in this figure. This is what prevents an even contact with the flat bottom surface from occurring.

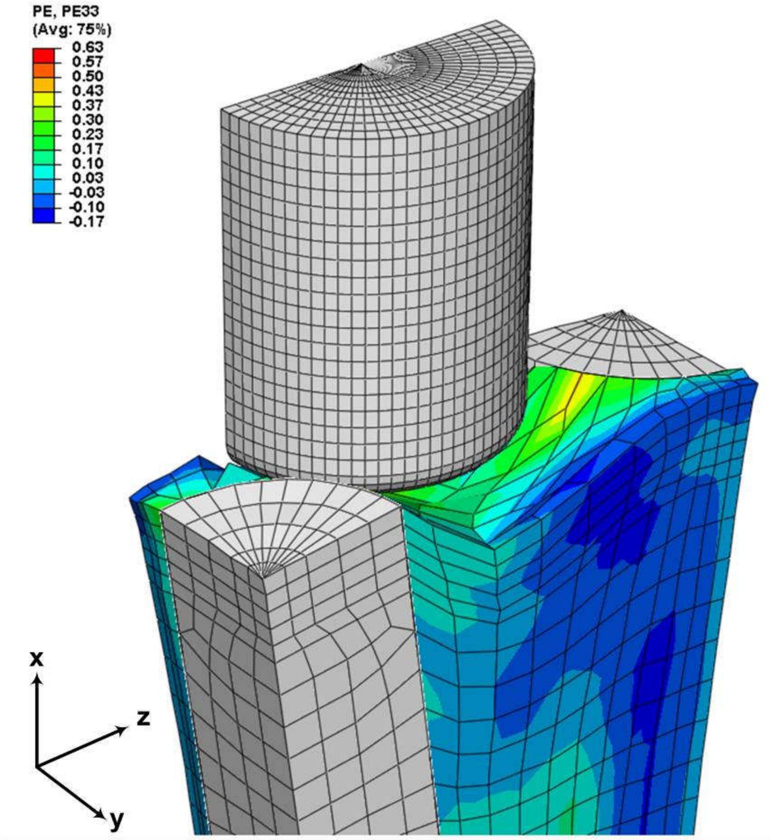

Figure 7: Plastic strain in matrix of deformed 3D unit cell model.

As with the 2D models the stress is the average value taken at the top end of the penetrating fibre and averaged over the area of the unit cell, while the strain is the end displacement divided by reference length L= 10 fibre diameters. The actual value in a laminate would be double as the wedged fibres would also be penetrating matrix material surrounding the penetrating fibre in this model. This was not included in this model to simplify the problem and reduce the number of elements in the model.

4.3 Two dimensional fibre cluster model

The two dimensional fibre cluster model is a larger version of the 2D single fibre model, with 10 fibres with symmetrical boundary conditions applied to the top and bottom of the model, (to replicate the model being part of a larger cluster). The length of the fibres is also much longer (100 fibre diameters), as this is the length required for kink bands to form, [5].The penetrating fibres are also a cluster identical to the cluster on the left hand side, rather than a single fibre as in the previous 2D model, Fig. 8.

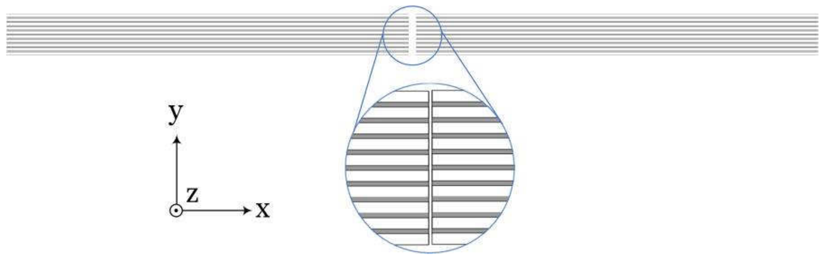

Figure 8: 2D Fibre cluster model

The concept of this model is to determine whether fibre kinking, chevron kinking or penetration is likely to occur, for the ideal fibre spacing used in the 2D model. A follow up study to this work will consist of a parametric study looking at irregular spacing of the fibres, Fig. 9a and different matrix materials, as well as Eccentricity of the broken fibres, Fig. 9b, irregularity of the fibre fracture crack surface, Fig. 9c and the initial waviness of the fibres represented in the form of a periodic sinusoidal wave.

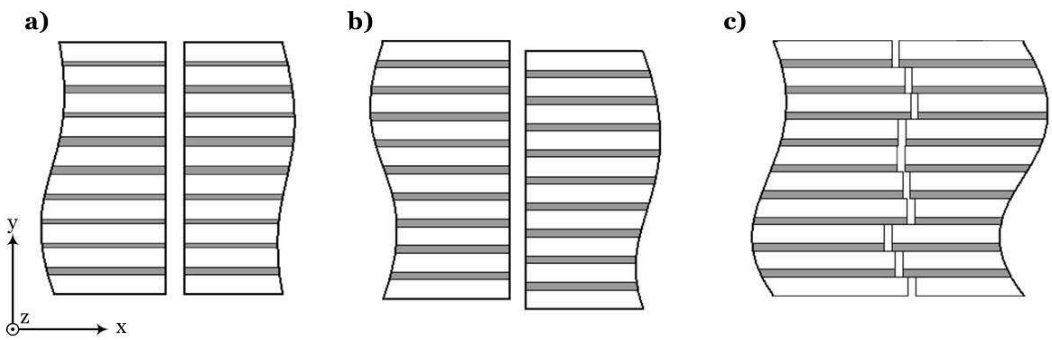

Figure 9: a) Irregular fibre spacing b) Fibre Eccentricity c) Fracture surface irregularity

5. CONCLUDING REMARKS

In conclusion this study is looking at a possible failure mode for impact damage composites that has not yet been considered in the literature, to the best of the authors' knowledge. The behaviour of the matrix material requires a fairly complex model which needs the input parameters to be carefully calculated and understood. The 2D models have produced reasonable results and are suitable for conducting parametric studies with relatively quick solution times, although a larger fibre cluster model would be required to capture all aspects of the failure mode and to consider the possibilities of fibre kinking or fibre chevrons. The 3D model should capture the stress strain relationship more closely than the 2D models but the large deformations in the matrix result in numerical instabilities causing the model to fail prematurely which needs to be addressed. Further refinement and parametric studies also need to be conducted, including consideration of friction and cohesive elements to allow for fibre matrix debonding.

ACKNOWLEDGEMENTS

This project is funded by EPSRC in UK under grant GR/T18783/01 with CASE support from Airbus UK. The authors would like to thank Dr. Jesper Ankersen for advice on FE modelling and Mr. Shashikant Pindoria for working on the models as part of a MEng final project in the Department of Aeronautics at Imperial College London.

REFERENCES

1- Davies, G. A. O. and Olsson, R., "Impact on Composite Structures", *The Aeronautical Journal*, 2004;108(1089):541-563.

2- Craven, R., Sztefek, P., and Olsson, R., "Investigation of Impact Damage in Multi Directional Tape Laminates and its Effect on Local Tensile Stiffness", *Composites Science and Technology*, 2008; Submitted for publication.

3- Fleck, N. A., "Compressive Failure of Fibre Composites" in *Advances in Applied Mechanics*, Academic Press, 1997:43-119.

4- Rosen, B. W., "Mechanics of Composite Strengthening", in *Fibre Composite Materials: Papers presented at a Seminar of the American Society for Metals October 17 and 18, 1964*, American Society for Metals, 1965:37-75.

5- Vogler, T. J., Hsu, S.-Y., and Kyriakides, S., "On the initiation and growth of kink bands in fibre composite. Part II: analysis", *International Journal of Solids and Structures*, 2001;38(15):2653-2682.

6- Olsson, R., Asp, L. E., Nilsson, S., and Sjögren, A., "A Review of Some Key Developments in the Analysis of the Effects of Impact Upon Composite Structures" *Composite Structures: Theory and Practice* ASTM STP 1383, ASTM, 2000;12-28.

7- Shivakumar, K. N. and Whitcomb, J. D., "Buckling of a Sublaminate in Quasi Isotropic Composite Laminate", *Journal of Composite Materials*, 1985;19(1):2-17.

8- Suemasu, H., Kumagai, T., and Gozu, K., "Compressive Behaviour of Multiply Delaminated Composite Laminates Part 1: Experiment and Analytical Development", *AIAA Journal*, 1998;36(7);1279-1285.

9- Suemasu, H., Sasaji, W., Aoki, Y., and Ishikawa, T., "Compressive Behaviour of Impact Damaged Laminates", *16th International Conference on Composite Materials*, ICCM 16, Kyoto, Japan, 2007:1-8.

10- Zeng, L. and Olsson, R., "Buckling Induced delamination Analysis of Composite Laminates with Soft Inclusion", *FOI-R--0412--SE*, FOI, Stockholm, 2002:1-34.

11- Dean, G. and Wright, L., "An Evaluation of the use of finite element analysis for predicting the deformation of plastics under impact loading", *Polymer Testing*, 2003;22(6):625-631.

12- Fleck, N. A. and Shu, J. Y., "Microbuckle Initiation in Fibre Composites: A Finite Element Study", *Journal of the Mechanics and Physics of Solids*, 1995;43(12);1887-1918.

13- Pinho, S. T., Darvizeh, R., Robinson, P., Schuecker, C., and Camanho, P. P., "Material and structural response of polymer-matrix fibre-reinforced composites", *Second World-Wide Failure Exercise*, 2008; Accepted for publication.