Dynamic delamination modelling using interface elements

L. Iannucci

Imperial College, Department of Aeronautics, Room 354B Roderick Hill Building, London SW7 2AZ, UK

Received 7 October 2004; accepted 1 February 2006

Abstract

Existing techniques in explicit dynamic Finite Element (FE) codes for the analysis of delamination in composite structures and components can be simplistic, using simple stress-based failure function to initiate and propagate delaminations.

This paper presents an interface modelling technique for explicit FE codes. The formulation is based on damage mechanics and uses only two constants for each delamination mode; firstly, a stress threshold for damage to commence, and secondly, the critical energy release rate for the particular delamination mode. The model has been implemented into the LLNL DYNA3D Finite Element (FE) code and the LS-DYNA3D commercial FE code.

The interface element modelling technique is applied to a series of common fracture toughness based delamination problems, namely the DCB, ENF and MMB tests. The tests are modelled using a simple dynamic relaxation technique, and serves to validate the methodology before application to more complex problems.

Explicit Finite Elements codes, such as DYNA3D, are commonly used to solve impact type problems. A modified BOEING impact test at two energy levels is used to illustrate the application of the interface element technique, and it’s coupling to existing in-plane failure models. Simulations are also performed without interface elements to demonstrate the need to include the interface when modelling impact on composite components.

© 2006 Elsevier Ltd. All rights reserved.

Keywords: Finite elements; Composite failure; Delamination modelling; Impact

1. Introduction

Laminated composites have been successfully used in a wide range of applications, ranging from large-scale use, such as aerospace and racing cars, to small-scale use, such as golf clubs and artificial limbs. The use of laminated composites is still increasing in these industries, this has been fuelled by the increasing sophistication of modelling tools based on Finite Element techniques. However, as the composite designer becomes more ambition in his/her designs, the corresponding modelling tools must also predict and simulate more complex and detailed scenarios/load cases, some of which may be non-linear and ill conditioned.

The design of composite components or structures to withstand impacts or crash is conceptually a difficult task for the composite designer. Unlike metallic components, which can yield and dissipate energy via plasticity, composites can only dissipate energy by a variety of damage modes, which usually degrades the stiffness of the structural component. Hence a reliable predictive tool is essential.

Laminated composites can fail in a variety of failure modes, typically matrix cracking, compression failure, fibre fracture, fibre kinking, and delamination between adjacent plies. The main inherent weakness of laminated composites is the extremely low through thickness strength, which is an obvious weakness within the composite meso-structure. This weakness can lead to interior delaminations when subjected to external loadings that generate high peeling stresses, namely the two through thickness shears and the normal stress perpendicular to the laminae. The maximum through thickness shear stresses generally occur in the mid-section of the laminate and drive the mode II and mode III...
delamination process. The mode III delamination can only occur in the bulk of the laminate. These internal delaminations can seriously reduce the compressive strength of the composite.

Delaminations can be caused by stress concentrations, edge effects, thermal curing stresses, impact, etc. and are usually given the term Barely Visible Impact Damage (BIVD) in the aerospace industry, when no in-plane damage is visible. The principal concern in aerospace are delaminations formed by impacts of varied objects, which can be caused by high velocity small mass, such as shell fragments, runway debris, stones, etc. or low velocity large mass such as tool drops. Depending on the velocity the structure or component may deform in a local or global manner and can affect the propagation of these delaminations. The impact energy is usually specified in the relevant Airworthiness Regulations; however, this energy can be composed of either high velocity/low mass or low mass/high velocity. Hence the material (rate dependency) and structural behaviour may differ for the same energy level. This inconsistency may be investigated if a robust delamination modelling tool is available to the composite designer.

The first step, can delamination initiation, propagation, and energy dissipation be modelled correctly. This paper covers the development on the BRITE ADCOMP [1] project of a dynamic delamination algorithm based on a damage mechanics approach, which has been implemented into the dynamic LLNL DYNA3D [2] FE code and the commercial LS-DYNA3D FE code (in the following text DYNA3D refers to both FE codes, unless otherwise stated). The proposed algorithm has been used to model a series of fracture toughness tests, ranging from pure mode I (DCB) to pure mode II (ENF), and including a mixed mode case (MMB).

The objective is to initially validate the proposed delamination algorithm against simple static delamination tests. The DYNA3D code can be used to model static problems in a pseudo-static or ‘dynamic relaxation’ fashion. Typically the inertia forces must be significantly smaller than the forces generated from the stiffness of the component or structure; otherwise spurious unreal modes are generated.

The new interface element is also coupled to existing composite in-plane failure models available within the DYNA3D code and used to model a series of impact tests available in the open literature.

2. Interface modelling

The ultimate strength of laminated composites containing stress concentrations can be predicted using numerical, analytical or semi-analytical methods. For the problems of in-plane cracks (occurring between plies) in a fibre composite, typically termed delaminations, the most widely used approaches are based on fracture mechanics or strength-based failure criteria. The fracture mechanics method uses the assumption that the delamination is effectively an edge crack occurring at the interface and thus it determines the stress intensity factor, \( K \), or the strain energy release rate, \( G \). A survey of fracture mechanics based approaches was performed by Garg [3]. For delaminations, which occur under the influence of shear and normal stresses, a strength of materials approach can be used to determine the onset of delamination. A review of failure criteria under impact conditions and based on the material strengths is given by Abrate [4]. Comparison of the two approaches, by Garg [3], indicates that the fracture mechanics approach is more accurate. However, the failure criteria based approach can only specify conditions under which potential cracks can appear they cannot give information about the length of such potential cracks. Fracture mechanics on the other hand cannot predict the growth of these cracks, if their length is unknown. Recent publications indicate a growing awareness of the application of a damage mechanics approach to explain and predict delamination damage in composites [5].

The damage mechanics approach originally developed by Kachanov [6] provides a method, which can potentially determine the full range of in-plane deterioration of a composite material, from the virgin material with no damage, to the fully disintegrated material with full damage [7–9]. In addition the method has the potential to predict different composite failure modes, and can provide a dissipation mechanism due to the formation of microcracks and micro-voids within the composite. The importance of post-failure mesh dependency has also been reviewed [10].

Traditional interface element modelling relies on a single specific element [5,8,11–14] (interface), which is homogenised and connects two adjacent ply layers. The approach has also been linked to ply failure models to simulate damage in open hole tests [15,16]. The original approach proposed by Crisfield et al. [13] for implicit techniques has also been implemented into the PAM-CRASH explicit code, and linked to in-plane failure models [17]. However, this approach does not guarantee failure of all inter-laminar modes simultaneously. Classical lamination theory is sometimes used to model the laminate above and below the interface, however, near edges or defects a full 3D computation must be performed, giving the three direct and three shear stresses. Existing interface elements are usually defined as having a ‘zero’ thickness, however, they must use an interface stiffness within their formulation which has a unique non-zero thickness, \('e'\), associated with the resin rich layer.

Delamination within the DYNA3D codes can be modelled using either a quadratic initiation criterion [3] or a tied node release algorithm based on a quadratic stress resultant. Recent versions of DYNA3D now offer control of the post-failure behaviour within the contact logic to mimic the mode II fracture energy required to create new fracture surfaces. Both approaches can predict the initiation of delamination, but will generally over-estimate the growth as no energy or power constraints are employed.
3. Proposed interface modelling strategy

The material model developed to describe the observed delamination damage is macroscopical. A macroscopical model describing every single microcrack is utterly unfeasible due to the large number of microcracks, which coalesce into a macrocrack or delamination. A mathematical approach to failure prediction has been developed which gives a measure of the average microscopic damage in the material. The measure is considered an additional internal state variable, and included as part of the constitutive relations for the material. Hence, in addition to strain, entropy, and temperature, the microscopic damage for a particular microscopical void is also specified. This approach encompasses fracture kinetics, which involves microscopic behaviour and can, if necessary, introduce specific rate dependence into the constitutive behaviour. This is most effectively modelled using a damage-lag approach, however, in the current formulation no rate enhancement in properties is assumed or modelled. This principle can only be applied if the damage is composed of a very large number of minute cracks, which are more or less evenly distributed throughout a volume element. The damage mechanics methodology must be based on a representative volume, unlike fracture mechanics. This means that the fracture process must be automatically built into the stress–strain relationship for the material, i.e., at a resin rich layer (volume). To prevent mesh dependent behaviour the constitutive relationship must be implemented as a stress relative-displacement relationship. The area under such a curve is equivalent to the facture energy.

Two fundamental relationships for each damage variable (mode I, II, III), which can be measured experimentally, should be:

- a stress threshold for damage to commence, typically the strength of the interface or resin,
- the area under the stress–strain curve should be equal to the energy dissipated, typically $G_i$ in the mode I delamination case.

And should also incorporate:

- A gradual reduction in stiffness. Typically as $d_t$ increases, the corresponding elastic moduli must also decrease.
- Gradual energy dissipation as damage develops. Energy must be removed from the system as cracks open.
- Displacements due to crack opening are smeared over an element (interface element). This interface element has a physical volume. Typically individual macrocracks are not modelled.

Details of the interface model may different for various types of composite, i.e., Z-pinned, NCF, etc. but for one particular composite material the model should be independent of the geometry and boundary conditions of the structure. If the model has been validated by a sufficient number of laboratory tests it should be applicable also to other structures made of the same material.

4. Numerical model formulation

The following section describes the interface element and its implementation into the DYNA3D code. An overview of relevant points used within the DYNA3D code is provided in the following section. For further details the LS-DYNA3D theoretical manual is highly recommended [18] and provides detailed examples of the methodology used within the code.

4.1. DYNA3D solution overview

DYNA3D is a non-linear material and geometrical explicit time integration Finite Element code. The code uses an updated Lagrangian formulation and a central difference time integration procedure [19]. The time-step is governed by the so-called Courant limit. Due to the explicit nature no stiffness matrix inversions are performed during an analysis. The time step is conditional stable, i.e., the shortest distance between two nodes within the FE mesh controls the time step. This can result in time steps of the order of $10^{-8}$ s, e.g., when modelling each individual composite ply with one solid finite element. However, with the use of Rayleigh damping and modification to the density, in cases where the inertia effects are negligible, a solution to a non-linear static problem can be obtained in a realistic time. Typically, the kinetic energy must be less than 0.001% of the total energy. This is sometimes referenced to as dynamic relaxation (DR).

The general equation of motion to be solved in DYNA3D is defined as

$$Ma^n = P^n - F^n + H^n$$

where $M$ is the diagonal mass matrix, $P^n$ is the external body loads, $F^n$ is the stress divergence vector, $H^n$ is the hourglass resistance, $n$ indicates the $n$th time step.

Rayleigh damping for non-linear static problem can be introduced by the additional of an appropriate damping matrix as shown below (2). The starting point is the dynamic equilibrium equations with the addition of a damping term.

$$Ma^n + C\dot{a}^n = P^n - F^n + H^n$$

where $C^n$ is the damping matrix.

Using the standard Rayleigh damping formulation, the mass and stiffness proportional damping can be introduced using the following equation:

$$C = \alpha M + \beta K$$

where $C$, $M$ and $K$ are the damping, mass and stiffness matrices, respectively. The constants $\alpha$ and $\beta$ are the mass and stiffness proportional damping constants, respectively. As recommended in the LS-DYNA3D manual a value of 10% was used for the $\beta$ in the high frequency domain.
The $z$ constant is set to critically damp the lowest frequency in the problem.

To advance to time $t^{n+1}$ the DYNA3D code uses a central difference time integration scheme, defined as

$$a^n = M^{-1}(P^n - F^n + H^n)$$

$$v^{n+1/2} = v^{n-1/2} + a^n \Delta t$$

$$x^{n+1/2} = x^n + v^{n+1/2} \Delta t$$

$$\Delta x^{n+1} = \frac{(\Delta t^2 + \Delta x^{n+1})}{2}$$

where $a$, $v$ and $x$ are the global nodal accelerations, velocities and displacement vectors, respectively.

Stress update during the time integration is trivial integrated incrementally, however, material rotation is considered using a Jaumann stress rate approach which ensures a co-rotational formulation and accounts for large movement of the local system with respect to the original position.

The treatment of sliding and impact is one of the major strengths of the DYNA3D family of explicit codes. Two methods are available within the code, firstly, a kinematic constraint method, and secondly a penalty method.

The kinematic method is explained in detail within the LS-DYNA3D theoretical manual [13]. Fundamentally, transformations are imposed on the nodal displacements of the slave nodes along the contact surface, thus eliminating the normal degree of freedom. The transformation imposes constraints on the global equations of the nodes along the contact surface. The second penalty method, which was used in the current study, places normal interface springs between all interpenetrating nodes and contact surfaces. Momentum is exactly conserved without the necessity of imposing impact and release specifications.

4.2. Kinematics

As discuss in the previous section the interface element formulation is embedded within a standard single Gauss point integrated solid element [20]. A local co-ordinate system is also established at the Gauss point of the element. The local direction is taken as the mean of the lamination thickness.

The local direction is taken as the mean of the lamination thickness. The transformation of the local system with respect to the original

4.3. Constitutive model

In the local co-ordinate system the stresses associated with the interface can be defined as

$$G_{13}, G_{23}$$ and $E_{33}$ are shear and Young’s moduli, respectively, of an ideal layer of thickness $e$ representing the interface. For numerical applications $G_{13}$, $G_{23}$ and $E_{33}$ can be assumed to be equivalent to a homogenised layer of the composite or to the values attributed to the matrix, $e$ can be assumed to a fraction (usually one-fifth) of a laminae layer thickness.

Following traditional interface element formulation, three damage variables are introduced $d_1$, $d_{II}$ and $d_{III}$ which are scalars and vary between 0 (no damage) and 1 (fully damaged), where $k_1$, $k_{II}$ and $k_{III}$ are the appropriate interface stiffness.

Fig. 1. Interface element characteristics.

Consider the constitutive relationship at the interface, which relates the stresses (or tractions) to the relative-displacements, Eq. (8).

$$\sigma_i = k_i(1-d_i)u_i$$

where $d_i$ ($i=I$, II, III) are the damage variables, and in incremental form,

$$\Delta \sigma_i = k_i(1-d_i)\Delta u_i - \sigma_i \frac{\Delta d_i}{(1-d_i)}$$

The increment of work dissipated for an increase in damage $\Delta d_i$ is,

$$\Delta W_i = \frac{1}{2}(\sigma_i \Delta u_i - u_i \Delta \sigma_i)$$

or using the incremental stress definition,

$$\Delta W_i = \frac{\sigma_i^2}{2k_i(1-d_i)^2} \Delta d_i$$

This is illustrated in Fig. 2. The damage energy release rate, which can be thought of as the equivalence of the fracture mechanics energy release rate $G_i$, is usually defined as $\frac{\Delta W_i}{\Delta d_i}$ and is sometimes given the symbol $Y_i$ [5]. Hence,

$$Y_i = \frac{\sigma_i^2}{2k_i(1-d_i)^2}$$
This is equivalent to
\begin{equation}
Y_i = \frac{1}{2} k_i u_i^2
\end{equation}
i.e., the apparent undamaged strain energy of the interface. The relative displacement can be made a function of this damage energy release rate, Eq. (15).

\begin{equation}
u_i = \sqrt{\frac{2}{k_i}} (Y_i)^{1/2}
\end{equation}

Damage evolution is sometimes made a function of the damage energy release rate divided by a critical value \cite{12}. However, it is clear from Eq. (15) that the relative-displacement and the damage energy release rate are related in a trivial fashion. Hence in the present formulation, damage evolution is made a function of the relative-displacement within a unit volume of material.

The evolution of damage follows a simple bilinear relationship given by

\begin{equation}
d_i = \frac{u_{\text{max},i}}{(u_{\text{max},i} - u_{0,i})} \left[1 - \frac{u_{0,i}}{u_i}\right]
\end{equation}

where \( i \) can be used to represent mode I, II or III, and \( u_{\text{max},i} \) is the strain at zero stress or damage = 1 (propagation), and \( u_{0,i} \) is the relative crack opening displacement at maximum stress or damage = 0 (initiation). The only parameters required for this evolution model are these two relative-displacement constants, which define the total energy dissipated, i.e., the area under the stress relative-displacement curve. Eq. (16) can be trivially converted into an incremental form, which has been implemented into the DYNA3D code.

\begin{equation}
\Delta d_i = \frac{u_{\text{max},i}}{u_{\text{max},i} - u_{0,i}} \left[\frac{u_{0,i}}{u_i}\right] \Delta u_i
\end{equation}

Hence the cumulative damage in the time domain is given by

\begin{equation}
d_i^{n+1} = d_i^n + \Delta d_i^{n+1}
\end{equation}

The bilinear relationship for mode I is shown in Fig. 3. Under compressive loading the interface element behaves a simple elastic material with the penalty stiffness based on the Young’s modulus. Unloading from a point \( A \) will always return to the origin, line OA in Fig. 3. Fig. 4 shows the shear behaviour in mode II and mode III.
The increment in work dissipated can also be defined in terms of relative-displacement as

\[
\Delta W_i = \frac{1}{2} k_i u_i^2 \Delta d_i = \frac{1}{2} \frac{u_{\text{max},i}}{u_{\text{max},i} - u_{0,i}} \sigma_0, i \Delta u_i
\]

(19)

In the time domain the total energy dissipated can be defined as

\[
W_{n+1} = W_n + \Delta W_{n+1}
\]

(20)

where \( n \) represent the current time step or cycle in explicit calculations. The total energy dissipated \( W_{n+1} \) is equal to the energy dissipated in creating the fracture surface. At failure \( W_i = G_{IC} \) when all the energy has been consumed.

To extend the formulation into a general mixed mode situation suitable for explicit dynamics, it is useful to introduce a series of dimensionless constants and variables. These constants and variables are defined below

\[
x_i = \frac{u_{\text{max},i}}{u_{0,i}}
\]

(21)

and

\[
\beta_i = \frac{u_i}{u_{0,i}}
\]

(22)

where \( i = I, II, III \) and takes the form for each delamination mode. Hence Eq. (16) now reduces to the following form:

\[
d_i = \frac{x_i}{(x_i - 1) \left( \beta_i - 1 \right)}
\]

(23)

when \( \beta_i = 1 \) damage is equal to 0 or initiation and when \( x_i = \beta_i \) damage is equal to 1 or propagation, i.e., complete de-cohesion has occurred at the interface for this mode.

The threshold stress for damage initiation was taken as 57 MPa [13] for mode I, and 100 MPa for the shear modes, which are typical values for a thermoset based epoxy system. Physically the initiation stress relates to the formation of microcracks, which ultimately leads to a delamination, once the critical energy has been consumed to create the fracture surfaces. If this information is not available, the through thickness strengths could be used as an estimate for these damage initiation stresses.

A single element test was performed to illustrate the behaviour of the element. The stress-displacement damage is shown in Fig. 5. In this case, the strain is normalised, i.e., the crack opening displacement, and the loading is a simple monotonically increasing relative-displacement. Based on available experimental data the simple model using a linear-damage law, based on traction, appears to work well. The only parameters required are the failure or initiation stress and the critical energy release rate. Convergence and rounding errors have been tentatively investigated. Conclusions indicate that the default time-step may need to be reduced by an order of magnitude to limit this type of error, if a coarse mesh is used. An iterative procedure may well be required towards the end of the damage process, especially when the dominant loads are force based and hence potentially unstable crack growth may be expected. Damage only commences at the threshold or initiation stress, which does not necessarily correspond with the statically measured resin strength. However, this was found to be fairly insensitive to the predicted results if the critical energy release rate remained constant. But can be more significant for pure mode II fracture problems such as may occur during impact.

Damage delay concepts can be introduced at this stage to modelled strain-rate effects, typically by introducing a time delay in the calculation of an increment in damage with respect to the stress. This requires Eq. (16) to be a function of time [21].

4.4. Mixed mode formulation

For mixed mode loading situations a strongly coupled approach is adopted, hence an effective damage parameter is postulated. The starting point is the well-known mixed mode criterion developed by Reeder [22].

\[
1 = \left[ \frac{G_I}{G_{IC}} \right]^{\lambda} + \left( \frac{G_{II}}{G_{IIIC}} \right)^{\lambda} + \left( \frac{G_{III}}{G_{IIIIC}} \right)^{\lambda}
\]

(24)

where \( \lambda \) is usually between 1 and 2.

Following the procedure outlined by Crisfield et al. [13] in converting to strain (relative-displacement) space. The fracture energy is defined as

\[
G_i = \int \sigma_i \text{d}u_i
\]

(25)

where \( i = I, II, III \).

Using Eq. (25) it is possible to reduce the energy equation to a strain based equation, Eq. (26), for point A in Fig. 6.

\[
1 = \left( \frac{u_i^{\alpha}}{u_{\text{max},i}} \right)^{2\lambda} + \left( \frac{u_{II}^{\alpha}}{u_{\text{max},II}} \right)^{2\lambda} + \left( \frac{u_{III}^{\alpha}}{u_{\text{max},III}} \right)^{2\lambda}
\]

(26)
This is taken as a starting point for the following equation:

\[ b^n_e = \left( \frac{u^n_I}{u_{0,I}} \right)^{2k} + \left( \frac{u^n_{II}}{u_{0,II}} \right)^{2k} + \left( \frac{u^n_{III}}{u_{0,III}} \right)^{2k} \]

(27)

where \( n \)th is the current time-step or cycle. Damage initiation corresponds to \( b^n_e = 1 \) and damage propagation to \( b^n_e = u_{\max} \). Since damage cannot decrease it is possible to specify a series of condition in time. Consider an increment in time for Eq. (27). The possible change in \( b^n_e \) can be given by

\[ \Delta b^n_{e+1} = b^n_{e+1} - b^n_e \]

(28)

If \( \Delta b^n_{e+1} > 0 \) use current value of \( b^n_{e+1} \) to replace \( b^n_e \).

If \( \Delta b^n_{e+1} < 0 \) unload along elastic slope, damage cannot heal hence \( b^n_{e+1} = b^n_e \).

If \( \Delta b^n_{e+1} = 0 \) stationary point.

Hence the effective damage can be given by the following equation:

\[ d^n_e = \frac{\alpha_e}{(\alpha_e - 1)} \left[ \frac{b^n_e - 1}{b^n_e} \right] \]

(29)

where

\[ \alpha_e = \frac{u_{\max,e}}{u_{0,e}} \]

(30)

and

\[ b^n_e = \frac{u^n_{e}}{u_{0,e}} \]

(31)

To illustrate the above approach it is possible to monotonically increase the relative-displacement with a fixed ratio maintained between \( u_I \) and \( u_{II} \) this is shown in Fig. 7. To facilitate damage in all modes simultaneously it is necessary for the following constraint to be enforced:

\[ \alpha_I = \alpha_{II} = \alpha_{III} = \alpha_e \]

(32)

This can be easily accomplished if the penalty stiffness in mode I is the only penalty input variable, then the corresponding penalty stiffness for mode II and III can be determined assuming the constraints of the bilinear curves follow Eq. (32). During damage evolution it is also necessary for

\[ d_I = d_{II} = d_{III} = d_e \]

(33)

if damage is less than 1. Otherwise:

\[ d_I = d_{II} = d_{III} = d_e = 1 \]

(34)

The damage or mode definition direction is aligned with the local material axis in the element, i.e., in the fibre direction for a 0/0 laminate. This can be set in the usual manner from within the DYNA3D code. In the example presented the mode ratio is maintained throughout the loading. Unloading for the mixed mode case is simply \( (1 - d_e) \) times the undamaged interface stiffness for the particular damage mode. Using Eq. (27) it can be seen that the mode ratio can change in time, such as what may occur during an impact or a crash event.

5. Fracture toughness simulations

The interface element described in the previous sections has been implemented into both the LLNL DYNA3D and the LS-DYNA3D codes. The implemented interface element was used to simulate a number of simple static fracture toughness tests using a dynamic relaxation approach, Table 1. This was considered a robust evaluation
of the interface element, to predict delamination initiation and propagation. A comprehensive set of delamination benchmarks were developed on the ADCOMP project and are now freely available for researchers to compare different fracture toughness modelling strategies [23]. The DCB benchmark is modelled in the following section and compared with the corresponding experimental test cases. Throughout the analyses the strain energy to kinetic energy is monitored to confirm the ratio remains to at least four orders of magnitude. The high frequency noise expected was not observed, this may be due to the viscosity coefficients used to control the unrealistic hourglass modes generated from single integration elements, or the shock-smearing algorithm within the explicit code.

The problems associated with softening materials are noticeably difficult to solve. A common problem with composites is the modelling of a brittle material with a stress-based failure. These types of failure models are common in explicit codes. Many researchers, notably Bazant [25,26] present mesh sensitivity studies, which indicate that, a crack can be made to propagate through a mesh with a lower load by refinement of the mesh. The use of a fracture energy type approach, whether implicit in the formulation or embedded within a constitutive relationship appears to be the ‘best’ solution. Bazant also discusses other techniques based on ‘non-local’ approaches and the use of characteristic lengths.

The damage mechanics formulation presented is concerned with the interface behaviour between laminated plies; three damage variables are introduced to model deterioration of the interface as damage develops. The formulation can use existing stress-based in-plane failure criteria for matrix and fibre damage [27,28]. Hence intra- and inter-laminar damage can be modelled within the same laminate. The current fracture toughness simulations assume no in-plane damage, but large displacements are considered in the formulation automatically. In the following sections in-plane damage is not considered, but is considered and coupled for the impact cases.

5.1. Mode I—double cantilever beam (DCB)

A standard DCB test, illustrated in Fig. 8(upper), was simulated [24] to demonstrate the use of the interface delamination algorithm embedded into DYNA3D. The Finite Element model (plane strain) is described in Table 1 and shown in Fig. 8(middle) and (lower). Interface elements are used to represent the resin rich area between the laminate arms. A displacement loading was applied at the free ends. The reactions at these points are automati-
cally calculated in DYNA3D and are plotted as a function of the crack opening displacement.

Fig. 9 illustrates the behaviour of the DCB specimen during the test. Also, in this figure are the test results and the analytically derived compliance and propagation curve [12,13]. The agreement between the test and the DYNA3D simulation results are very good, especially when only two frequencies could be critical damped to simulate this DCB pseudo-static problem. Differences between the analytical and numerical results would be expected, as the arms are modelled with solid elements, hence shear deformation is considered. Also, large displacement and co-rotational formulations are automatically considered with the DYNA3D code. To illustrate the functionality of the interface element approach, which is based on fracture mechanics, a comparison is made with the existing stress-based delamination algorithm in DYNA3D (within material model 22). This is shown in Fig. 10, which clearly indicates the importance of using fracture mechanics when modelling delamination.

The energy release rate can be computed directly using the conventional area method from this curve. Results clearly illustrate the excellent correlation using the new approach.

DCB testing between dissimilar ply orientation when fibre bridging and crack jumping occurs can make predictions of energy release rates very difficult. However, coupling the existing in-plane composite failure options (matrix and fibre) in the DYNA3D code with the improved delamination modelling technique, can potentially open the possibilities of modelling this problem, without specific user intervention during the simulation.

Fig. 8. Upper: DCB displacement loading (length \(2L = 100\) mm, \(a_0 = 30\) mm, \(2h = 3\) mm). Middle: DCB Finite Element mesh. Lower: DCB Finite Element mesh (close-up).

Fig. 9. DCB simulation using interface element technique and comparison with test and analytical solutions.
5.2. Mode II—end notched flexure (ENF)

A standard ENF test, Fig. 11(upper), was simulated to illustrate the mode II behaviour of the interface element. Finite Element model details are given in Table 1. Again the resin rich layer is modelled with the new interface elements. Fig. 11(middle) indicates the mesh used during the study in the partially loaded state. A displacement loading was applied at the free end as illustrated in Fig. 11(upper). The reactions at this point are automatically calculated in DYNA3D and are plotted as a function of the mid-point displacement. Fig. 11(lower) shows the deformed mesh with the distorted interface element. Unlike the mode I simulations the laminate arms must be allowed to slide. This is easily accomplished using the in-built DYNA3D penalty contact logic. The corresponding load displacement plots are shown in Fig. 12. Also in this figure the analytically derived compliance and propagation curve [12,13] for a mode II critical energy release rate of 4 N/mm. In addition, the results for an ENF simulation with a critical energy release rate of 1 N/mm. The modelling of bifurcation points within these load displacement responses, when using a very low critical energy release rate, can be very difficult to solve numerically with implicit techniques, even with arc-length methods [13]. However, the explicit DYNA3D method can easily predict the response, even with low energy release rates.

5.3. Mode III—mixed mode bending (MMB)

A standard MMB test [29] was simulated to illustrate the use of the interface delamination algorithm when under a mixed mode loading. Fig. 13(upper) illustrates the generic set-up for such a test. Details of the Finite Element model (plane strain) are shown in Table 1. The mixed mode interaction parameter \( \lambda \) had an assumed value of 2, which is
typical value for a thermoset composite [13]. The Finite Element mesh in the deformed state and a close-up of the propagation region are shown in Fig. 13 (middle) and (lower), respectively. A displacement loading was applied at the level end with $c = 43.7$ mm which corresponds to a mixed mode ratio of $G_I/G_{II} = 1$. The level arm is modelled

Fig. 12. ENF simulations and comparison with analytical solutions.

Fig. 13. Upper: mixed mode displacement loading (length $2L = 100$ mm, $a_0 = 30$ mm, $2h = 3$ mm). Middle: mixed mode Finite Element mesh. Lower: close-up of mixed mode propagation.
using shell elements with constraint equations applied, linking the dof of the laminate fixing and sliding points to the relevant level arm points. The reaction at the loading point on the level arm is automatically derived in DYNA3D and is plotted as a function of the crack opening displacement. Contact logic was not used to prevent unnecessary spurious noise from being generated at the level arm contact point.

The load displacement plot for the MMB and the corresponding analytical solutions are shown in Fig. 14. Again, the analytical derivations are based on the assumption of small displacement, and do not include effects from shear deformation, and large movement of the local material axis. In this case the local co-ordinate axis of the interface element requires a co-rotational formulation (this is included within DYNA3D) to model the MMB test for the full range of displacements. MMB corrections are discussed by Reeder and Crew [29], amongst many other researchers.

6. Impact modelling

The proposed damage mechanics approach to impact modelling de-couples the in-plane and out-of-plane failure criteria within the DYNA3D code. The embedded interface element is used to model delamination or de-cohesion between sub-laminates only. Thus the existing fibre and matrix failure criteria can still be used, but with the addition of an energy based delamination algorithm [30].

The in-plane failure for the case studies uses solid-shell elements, not solid elements. These types of elements do not suffer as severely from the mesh dependency problems discussed. Principally this is because the shell element formulation assumes plane sections to remain plane, and hence the problems associated with stress concentrations when using solid elements is eliminated.

The simulation described in the present paper covers the three-dimensional modelling of two impact load cases taken from the open literature [31].

6.1. Delamination modelling methodology

During a low speed impact event a number of damage mechanisms may occur, typically delaminations, matrix cracking and matrix compression, fibre fracture and fibre kinking. Fig. 15 illustrates a fractograph of a typical impact. The figure clearly shows that delaminations may initiation and propagate within each lamina, however, it is unrealistic to model each potentially debonding layer with Finite Elements, see inset in Fig. 15.

It is assumed that an envelope of all the delaminations at the mid-plane could be used to model the loss in flexibility of a laminate. Clearly, the maximum shear stress depends on the lamination angles, but will generally be located at or near the mid-plane of a laminate, hence this assumption should give a representative maximum envelope of the delamination damage map. Fig. 16 shows the approach adopted with the resin rich layer between two sub-laminates of solid-shell elements. The degrees of freedom (dof) match precisely. If shell elements were used, then constraint equations would be necessary to couple the rotational dof to the translational freedoms on the interface elements.

As the interface elements are volume based it is possible to model the resin layer adjacent to the upper and lower lamina, as delaminations will always tend to propagate along the upper and lower surface. This eliminates the need of using the mean of the upper and lower lamina system. A half interface (two interface elements) can be used to investigate and model delamination growth which occurs along fibre directions (upper and lower interfaces), the resolved stresses ahead of a delamination crack will dictate whether
the delamination will then grow along the upper or lower resin-fibre surface. Furthermore, as the delamination tends to grow along the fibre direction only the critical energy release for a 0/0 configuration will be required (the stiffer direction), the modelling of delamination ply jumping can be realised when a through thickness matrix failure criteria is also adopted.

6.2. Finite element models

A plan view of the Finite Element models is shown in Fig. 17. Table 2 gives a summary of the numerical models used. The impactor was modelled with solid elements, while the composite laminate was modelled with solid-shell elements. The solid-shell element formulation uses an existing stress-based (Chang–Chang) in-plane failure criteria for matrix and fibre damage initiation at a lamina level.

Two models were generated. Model A uses a basic solid-shell element mesh for the 1 mm composite laminate, while for model B the laminate is modelled with two sub-laminate meshes of solid-shell elements with the interface element between these two sub-laminates. For model B the delaminations are assumed to initiate and propagate at the mid-plane of the model, i.e., only a single layer of interface element is used within the FE model. The solid-shell use an in-plane failure model based on Chang–Chang [19,28]. All lamina within the composite laminate were modelled within the solid-shell elements. For the 1 mm composite a total of eight through thickness integration points were used to model all the UD lamina. A contact slide-line was placed between the impactor and the composite.

6.3. In-plane failure model

Non-linear explicit FE codes often have implemented in-plane failure models to predict impact damage. They can be based on traditional failure criteria, such as maximum strain, maximum stress, Tsai-Wu or Tsai-Hill criteria. These criteria are generally all in agreement for uni-axial failure stresses in the principle material direction of a lamina. The criteria differ on what constitutes failure for bi-axial stress states. Furthermore, these existing criteria
have not been validated in a dynamic situation, but are generally considered to be reliable predictions of the initiation of damage. The existing plane stress DYNA3D failure model for UD carbon composites is based on Hashin [27] and Chang–Chang [28] formulations with five material parameters used in the three failure criteria, namely:

\[
\begin{align*}
X_t &= \text{longitudinal tensile strength}, \\
Y_t &= \text{transverse tensile strength}, \\
S_c &= \text{shear strength}, \\
Y_c &= \text{transverse compressive strength}, \\
a &= \text{non-linear shear stress parameter},
\end{align*}
\]

where \(X_t\), \(Y_t\), \(S_c\), \(Y_c\) and \(a\) are obtained from material strength measurements, \(a\) is usually defined by material shear stress–strain measurements.

For plane stress conditions the stress–strain relationship can be defined as

\[
\begin{bmatrix}
\sigma_1 \\
\sigma_2 \\
\sigma_{12}
\end{bmatrix} =
\begin{bmatrix}
C_{11} & C_{12} & 0 \\
C_{12} & C_{22} & 0 \\
0 & 0 & C_{44}
\end{bmatrix}
\begin{bmatrix}
\varepsilon_1 \\
\varepsilon_2 \\
\gamma_{12}
\end{bmatrix}
\]

where

\[
\begin{align*}
C_{11} &= \frac{E_1}{1 - v_{12}v_{21}}, \\
C_{12} &= \frac{v_{12}E_2}{1 - v_{12}v_{21}}, \\
C_{22} &= \frac{E_2}{1 - v_{12}v_{21}}, \\
C_{21} &= \frac{v_{12}E_1}{1 - v_{12}v_{21}}.
\end{align*}
\]

The current DYNA3D failure model (Material Model 22) uses a fibre-matrix shearing term, which augments each damage mode.

\[
\bar{\sigma} = \frac{\sigma_{12}^2}{S_{12}} + \frac{a}{2} \sigma_{12}^4
\]

where \(\bar{\sigma}\) is the ratio of the shear stress to the shear strength.

The existing DYNA3D damage model implemented into the plane stress shell element considers three modes of damage:

1. fibre fracture,
2. matrix cracking,
3. matrix compression failure.

Each damage mode is associated with a criterion for failure, which is based on the stress in the lamina exceeding an appropriate strength.

**Fibre fracture failure criteria**

\[
F_{\text{fibre}} = \left(\frac{\sigma_1}{X_t}\right)^2 + \bar{\tau}
\]

failure is assumed whenever \(F_{\text{fibre}} > 1\). If \(F_{\text{fibre}} > 1\), then

\[
E_1 = E_2 = G_{12} = v_{12} = v_{21} = 0
\]

**Matrix cracking failure criteria**

\[
F_{\text{matrix}} = \left(\frac{\sigma_2}{Y_t}\right)^2 + \bar{\tau}
\]

failure is assumed whenever \(F_{\text{matrix}} > 1\). If \(F_{\text{matrix}} > 1\), then

\[
E_2 = G_{12} = v_{12} = v_{21} = 0
\]

**Matrix compression failure criteria**

\[
F_{\text{comp}} = \left(\frac{\sigma_2}{S_c}\right)^2 + \left(\frac{Y_c}{2S_c}\right)^2 - 1 + \frac{\sigma_2}{\bar{Y}_c} + \bar{\tau}
\]

failure is assumed whenever \(F_{\text{comp}} > 1\). If \(F_{\text{comp}} > 1\), then

\[
E_2 = v_{12} = v_{21} = 0
\]

However, the modelling approach does have some limitations. These are summarised below

- Results are element size dependent.
- Difficulty matching experimental data for compression failure.
- Numerical noise may cause premature failure.
Elastic response to failure is unrealistic.

Ad-hoc treatment of post-failure behaviour.

7. Results

The force time history curves for case I are shown in Fig. 18. The curves presented are the experimentally measured values [31]: the single laminate and the double laminate with interface element cases, respectively. The corresponding displacement time history curves are shown in Fig. 19. A comparison between model A and B for the kinetic energy of the impactor for case I are shown in Fig. 20. The model A response is almost elastic. This was confirmed by the symmetrical nature of the force and displacement time histories.

The force time history curves for case II are shown in Fig. 21. The curves presented are the experimentally measured values [31]: the single laminate and the double laminate with interface elements cases, respectively. The corresponding displacement time history curves are shown in Fig. 22. A comparison between model A and B for the

### Table 2

<table>
<thead>
<tr>
<th>Title of numerical model</th>
<th>Finite Element details</th>
<th>Test and material properties details</th>
</tr>
</thead>
</table>
| Impact case I | Impact Model A 45,665 nodes 16,200 solid-shell elements (laminate—2 through thickness) 19,728 solid elements (impactor) | Ref. [31] Impact energy = 0.67 J Impactor mass = 0.412 kg 0.5 inch diameter spherical TUP Laminate thickness 1 mm Lay-up [+45/−45/0/90/90/0/90/−45/+45] Standard BOEING size (100 mm×75 mm) $E_{11}$ (fibre direction) = 140 GPa $E_{22}$ (matrix direction) = 9.5 GPa $G_{12}$ (in-plane shear) = 5.8 GPa $\sigma_{11}$ (tensile strength) = 2000 MPa $\sigma_{22}$ (tensile strength) = 70 MPa $\sigma_{12}$ (compressive strength) = 1650 MPa $\sigma_{23}$ (compressive strength) = 240 MPa $\sigma_{13}$ (shear strength) = 105 MPa $\sigma_{22}$ (shear strength) = 105 MPa Damage initiation stress (mode I) = 57 MPa Damage initiation stress (mode II) = 100 MPa Damage initiation stress (mode III) = 100 MPa Interface $G_{IC1}$ = 0.281 N/mm Interface $G_{IC2}$ = 0.900 N/mm Interface $G_{IC3}$ = 0.900 N/mm Mode interaction parameter $\lambda = 2$
| Impact Model B—with interface elements 53,946 nodes 16,200 solid-shell elements (laminate—2 through thickness) 19,728 solid elements (impactor) 8100 interface elements (resin layer) | |
| Impact case II | Impact Model A 45,665 nodes 16,200 solid-shell elements (laminate—2 through thickness) 19,728 solid elements (impactor) | |
| Impact Model B—with interface elements 53,946 nodes 16,200 solid-shell elements (laminate—2 through thickness) 19,728 solid elements (impactor) 8100 interface elements (resin layer) | |
Fig. 18. Force–time plot for 0.67 J impact case.

Fig. 19. Displacement–time plot for 0.67 J impact case.

Fig. 20. Kinetic energy–time plot for 0.67 J impact case.
The kinetic energy of the impactor for case II are shown in Fig. 23. In the higher energy impact case it can be seen that the difference between the two models is significantly greater, this is because the delamination consumes energy.
and degrades the stiffness, thus allows further matrix cracks to develop, since the laminate is now more flexible.

The initial parts of the experimental traces show a marked disagreement with the numerically predicted force histories, although the displacement histories appear considerably more accurate. This was considered to be due to the data processing of the original raw data, which was recorded using an 8-bit transient recorder and was thought to have been excessively smoothed. The original raw data was not available, but similar data was investigated and confirmed this could account for the initial error.

The extent of the delamination damage map at the mid-plane predicted by the interface elements are shown in Fig. 24 for case I and case II. It was reported [31] that the case I impact generated no delaminations, however, the interface elements have predicted a small delamination (only at the mid-plane) zone with an initiation stress of 100 MPa based on the through thickness strength of the laminate. This was increased to 125 MPa, maintaining the fracture toughness value from the previous case; the simulation then predicted no delaminations. This is a relatively high value, but evidence has been found in the open literature for such values [33]. Furthermore, the modelling approach only assume the delamination to initiate and propagate along a single plane. If CPU time was not a constraint, then all such interfaces between layers could be modelled!

A simple analytical validation example can be used to check the threshold for mode II delaminations to commerce. A critical force exists for damage to propagate indeterminately in an elastic isotropic plate with a single damageable layer at the mid-plane [32]. The critical force is independent of crack (defect) size. The analytical derived critical force level is given by

![Fig. 24. Interface element delamination failure envelope. Upper: 3.11 J impact case. Lower: 0.67 J impact case.](image)

![Fig. 25. Fibre fracture plot on lamina furthest from impactor. Left case: single laminate with no interface element. Right case: two sub-laminates with interface element.](image)
\[ P_C^2 = \frac{8\pi^2E(2h)^3}{9(1 - \nu^2)} G_{\text{CII}} \]  

where \( P_C \) is the critical load for a mode II delamination to propagate, \( G_{\text{CII}} \) is the mode II critical energy release rate, \( E \) is the Young's modulus, \( \nu \) is the Poisson's ratio, \( 2h \) is the thickness.

Eq. (40) is based on the assumption of small displacements, and does not include effects from shear deformation, orthotropic materials and large movement of the local material axis. For the impact case the local co-ordinate axis of the interface element requires a co-rotational formulation, which is included within DYNA3D. Furthermore, for thin laminate (typically 1 mm) the peak displacement can be substantially greater than the thickness (especially under impact).

The critical force threshold for case I was approximately 800 N and corresponds to a displacement of approximately 2.4 mm. The same threshold force was also observed for case II. The predicted force threshold using Eq. (40) for the 1 mm case was 680 N. The differences between the numerically observed threshold force and the analytically determined value can be attributed to the basic assumptions used in the derivation of this equation. The peak experimental force for case I was 730 N and for case II 1100 N, which corresponds to a peak displacement of 2.4 mm and 5.2 mm, respectively. The peak displacement is in excess of the thickness of the laminate; hence membrane stresses in addition to the bending stresses are generated. This can result in the movement of the peak shear stress within the laminate.

No fibre fracture was predicted using the in-plane failure model for case I. The fringe plot for fibre fracture within the laminate for model A and B for case II in the outermost ply is shown in Fig. 25. The fibre damage patterns are very similar, however, the double laminate does show a slightly greater extent as would be expected in a more flexible laminate.

8. Conclusions

The methodology proposed in this paper reflects a novel technique that can be used to model a range of composite design problems, such as FOD, bird strike, fragmentation attack, ballistic damage, etc. where dynamic delamination is an important consideration.

Fracture toughness comparisons have been restricted to pseudo-static simulation of tests; results and analytical solutions are available in the open literature. Considering a highly non-linear dynamic code has been used to simulate static delamination problems the agreement is good. The formulation can allow rate dependency to be implicitly included within the constitutive relationship by the use of a visco-elastic or damage-lag approach, hence high rate fracture toughness tests could also be simulated and rate effects investigated.

In-plane degradation can be modelled by including an appropriate failure model within the solid or solid-shell laminates adjacent to the interface element. The interface element must be placed where delaminations are expected, hence some prior knowledge of their propagation path is required, if computing resources are limited. The major advantage is that no initial defect or starter cracks are necessary. Similarly the material constants can easily be derived from standard laboratory tests. The approach, being fracture based, does not suffer from the mesh dependency of stress-based approaches.

Two standard impact cases are considered and modelled, including both in-plane damage and delamination via the interface element approach. The in-plane and out-of-plane models are decoupled, i.e., coupling is implicit in the physics of the impact. A comparison of the experimental measured and the numerically calculated force-time history for impact case I and case II using model B are shown to be in very good agreement. This was initially surprising considering the ad-hoc treatment of in-plane failure within the DYNA3D code, however, a fine mesh was used hence the unloading (softening) response, which is based on 100 time-steps, is very steep. This is characteristically observed for UD type composites, also the use of a non-linear shear-stress term prevent fibre failure occurring prematurely as can occur when a linear shear-stress relationship is used. The delamination map is also in very good agreement with the experimentally observed damage map. A comparison of the force-time histories between model A and model B for case II indicates a differences of approximately 9% at the peak load. This difference is mostly associated with the energy required to propagate the delamination. Furthermore, using two sub-laminates with a single layer of interface elements adjacent to the sub-laminates, results in a more compliant laminate, this leads to greater in-plane damage during the impact event. The common assumption of plane sections remaining plane during damage growth may need to be re-visited, but this may cause stress-localisations problems. Initial indications show that flexural degradation is enhanced during impact when this assumption is relaxed, i.e., a greater in-plane damage map would be predicted.

The approach was extended in the ADCOMP project to the modelling of the residual compressive strength (CAI) of an impacted laminate. This was achieved by modelling the static compressive load as a second load case, which follows the impact case. This will be the subject of a second paper.

The out-of-plane formulation presented is relatively simple and can be easily implemented in the existing solid element based material model 22 in DYNA3D. Thus providing a fracture based delamination model within a very popular composite Material Model. However, the integration points for single point integrated elements, will be at the centre of the solid elements, rather than within resin rich layer between sublaminates.
The approach could allow a cradle-to-grave design methodology to be adopted, as the damage mechanics approach is based on a characteristic volume, the approach lends itself to damage initiation (or initial damage) from Non-Destructive-Evaluation (NDE) ADSCAN data (via a conversion program to initial damage). Further, the approach can be used to cumulatively add damage from a wide range of potential load cases.

Acknowledgements

The author acknowledges many useful discussions with Mechanical Science Sector (DERA, Farnborough, UK) staff and the contribution of the BRITE III ADCOMP project.

References


