J Mater Sci (2008) 43:6586–6592
DOI 10.1007/s10853-008-2695-x

STRETCHING THE ENDURANCE BOUNDARY OF COMPOSITE MATERIALS: PUSHING THE PERFORMANCE LIMIT OF COMPOSITE STRUCTURES

Virtual testing of realistic aerospace composite structures

G. A. O. Davies · J. Ankersen

Received: 31 March 2008 / Accepted: 30 April 2008 / Published online: 31 July 2008

Abstract The feasibility of using simulation of component tests to failure is examined from the view of using this as a design tool to reduce time and costs for future aircraft designs. All aspects are considered, that is modelling correctly the physics of failure mechanisms, having an acceptable CPU time, and ease of interfacing with existing CAE systems for generating models and processing the results. By looking at many benchmarks (including an industrial component) with experimental validation, it is concluded that simulation is now a feasible design tool and that many more searching examples will lend credibility to this strategy for both manufacturers and certification authorities.

Introduction

This paper addresses the problem of using simulation of composite structural behaviour as a design tool by industry. This is a somewhat different view from that of the common journal paper which tends to concentrate on getting the material physics correct or improving the computational algorithms. We have not ignored these aspects but have concentrated on the design-led features. The need for so doing is clear. Design time is expensive and structural testing also, of the order of $40 million for a new aircraft variant. The traditional way of designing and certifying modern aircraft, both civil and military, has been to use finite element codes as a means of finding the internal stress fields under key loading states, and then to use data sheets to deduce strength and failure modes. These data sheets for metallic structures have been based on a wealth of experience. In fact the extensive Airbus family has such a well established pedigree that a knowledge-based system has been compiled to design new aircraft. This is a very rapid design tool which is not possible for carbon composite structures which are usually radical and have very complex failure modes. Testing is necessary.

There are two extreme forms of testing undertaken. Firstly there may be thousands of small simple coupon specimens to establish basic laminate properties of stiffness, strength, and toughness. Secondly there will be one or two obligatory tests of the complete aircraft, or major component, to ultimate load or for fatigue life. However in between there are many tests rising from elements, to details, to components, and to sub-structures for which the internal stress fields have been thought too intricate to evaluate by simple analysis. The aim is to reduce the need for many of these tests. Figure 1 (with acknowledgements to BAESystems) summarises the challenge succinctly.

Modelling requirements

A typical structural test to failure is sketched in Fig. 2. Because fibres and matrices are essentially brittle in nature, it is possible that a very local stress concentration can cause damage which will then propagate in an unstable fashion and lead to complete failure. This is relatively simple to model with a linear analysis, but is hardly a fail-safe design. More desirable, and hopefully more common, is the second curve where an embryonic damage propagates in a stable fashion into (say) a decreasing stress field, for example delamination due to low velocity impact; or there

G. A. O. Davies (✉) · J. Ankersen
Department of Aeronautics, Imperial College, London SW72BY, UK
e-mail: g.davies@imperial.ac.uk

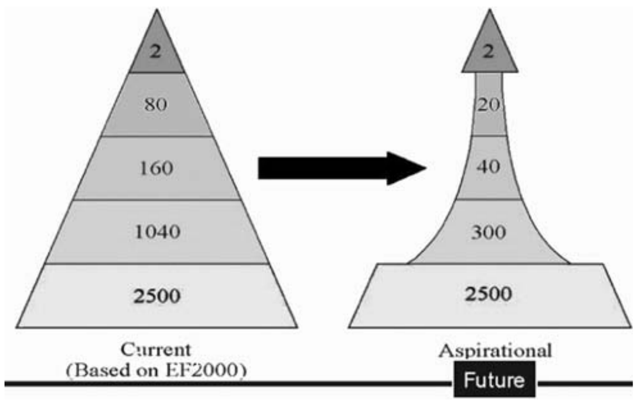


Fig. 1 Aim of reducing the number of component tests (courtesy of BAESystems)

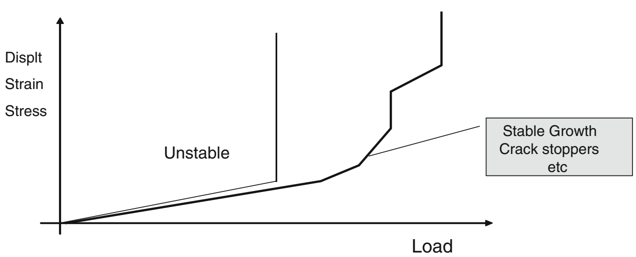


Fig. 2 Typical structural response during test to failure

may be a mode change in buckling, or crack jumping from one ply to another; or the damage may meet a crack stopper at some structural feature. Clearly any FE analysis now has to be nonlinear. At Imperial College we developed over many years such a code (FE77) to simulate the many complex failure mechanisms in laminated composite structures. However in recent years there has emerged commercially two popular front runners which do capture the physics, are robust, are relatively easy to use, and which can be coupled to an existing CAE code or pre/postprocessor such as PATRAN in the aerospace sector. LS DYNA3-D was developed over 25 years ago, initially for dynamics and crashworthiness. It therefore used as an explicit solver but is nevertheless even used nowadays by industry to simulate metal-forming processes. It now has an implicit version NIKE. The other, ABAQUS Standard, is now 30 years old and was initially an implicit solver with many special features appropriate to nonlinear behaviour. It also now has an explicit version. Because of our connections with BAESystems we followed their choice of ABAQUS in 2006, and we note that Airbus (UK, France, Germany.) have recently followed suit. The remainder of this paper uses ABAQUS as the simulation code.

Having an FE code is just the beginning of the solution. Even with friendly software, the modeller has to create an FE model which is capable of capturing the internal stress fields with the required accuracy. Traditionally this means the engineer views the output of stresses before smoothing, and then judges whether a finer mesh is needed. A second solution will confirm this. However in laminated composites, which may delaminate or debond between components, another factor has to be recognised.

If a crack develops, it has been shown that simulating its propagation needs an accurate representation of the stress field ahead of the crack front. This region, known as the "process zone" in metals [1] may be small, and needs to have finite elements small enough to simulate. A common and convenient estimate comes from the 2-D elastic strain field at the tip of a crack in an infinite space (rising to infinity at the crack tip) and then truncating it at a simple yield stress σ_y. The expression for the length of the plastic zone is then:

$$l = \frac{\pi E G_c}{8\sigma_y^2} \tag{1}$$

where E is the modulus and G_c is the mode I fracture toughness.

However in laminated composites, where delamination or debonding takes place, the crack is usually bounded by two thin plates of finite flexural stiffness. A model of two plates, separated by an elastic layer of resin (in Fig. 3) gives a different approximation in terms of plate modulus E and thickness d, and resin modulus E_R and thickness t.

$$l/d = \frac{\pi}{4}\left[\frac{1}{3}\frac{E}{E_R}\frac{t}{d}\right]^{\frac{1}{4}} \tag{2}$$

For typical values of these four measures we find that l is of order 1 mm. This unfortunate fact has been confirmed by an FE simulation for the standard double-cantilevered-beam (DCB) test for mode I fracture toughness. The four curves in Fig. 4 have been shown for two beam depths and two fracture toughness's. The need for such a small mesh size clearly implies a very local refined model.

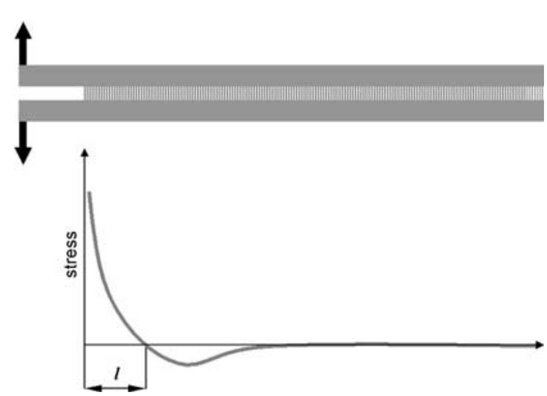


Fig. 3 Stress variation in resin between two flexible plates

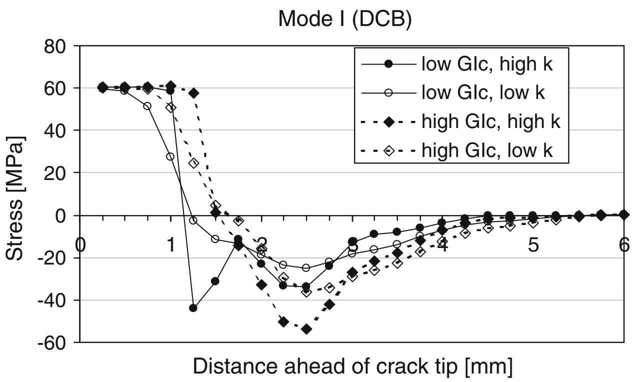


Fig. 4 FE confirmation of interface stress variation ahead of a crack in DCB

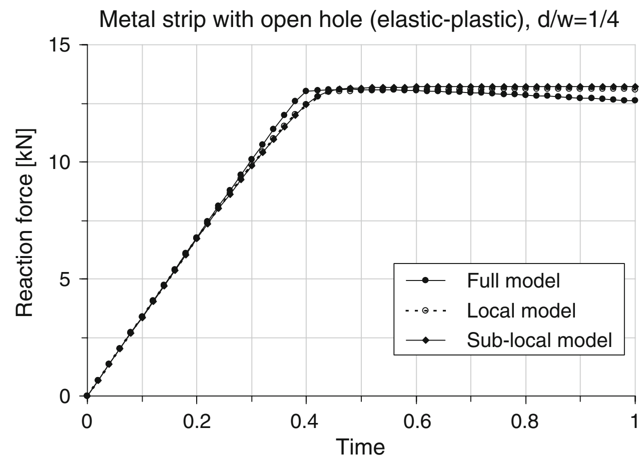


Fig. 6 Loading history up to failure in yield for three models

A local/global strategy

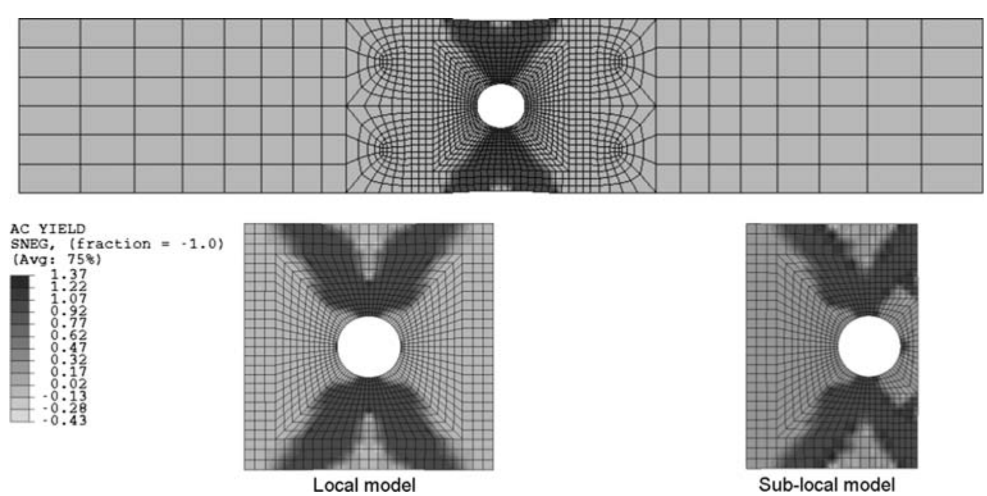
A mesh size of order 1 mm is clearly unacceptable for a complete aircraft, or even a modest component. Most codes now contain a local/global strategy whereby an existing coarse (global) finite element model can have embedded within it a refined (local) region. The choice is left to the user, based on the global stress field or just judgement of the local geometry. However there may potentially be many such sites of local stress concentration, and a non-linear analysis of the full structure, with a multitude of local regions, would be computationally prohibitive. A reasonable strategy is therefore to select each local region of a size (say) two or three times the expected size of the damage zone, and then apply the tractions/displacements at the local boundary up to failure, but without changing the pattern at the boundary. Each local model can then be analysed separately. No general proof of the validity of this strategy is possible (apart from an appeal to SaintVenant), but the following example in Fig. 5 gives confidence.

The figure shows a long strip of a ductile metallic material with a central circular hole. If we load the strip in tension, then local yielding starts at the hole edge and then spreads across the strip with increasing load, until complete failure. We might therefore expect the stress distribution across the width to change significantly as the strip yields. The first local model has boundaries placed 3 radii away from the hole centre, and the pattern of yield looks identical to the full model. The second sub-local model has a boundary only one radius away from the hole edge, and yet the yield pattern seems to be preserved. This is confirmed in the displacement history of the loaded ends, as shown in Fig. 6, where all three curves are almost identical.

The physics of material failure

As mentioned we will not embark on the plethora of models for simulating laminated composite material failure [2] but suffice to say that the following are now standard default strategies in ABAQUS.

Fig. 5 Metal strip subjected to tension loading to failure by yield



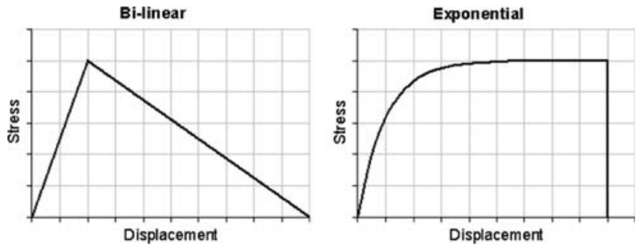


Fig. 7 Two types of interface stress/displacement law

Firstly delamination, or debonding between components, involves a crack with infinite stresses at the crack front in an elastic solution. A form of fracture mechanics is consequently needed to test the crack for propagation. Most codes have the virtual crack extension (VCE) or closure (VCC) method whereby the crack is opened and the strain energy release rate evaluated and put equal to the critical value of G_C for propagation. This is not an attractive computational strategy where there may be many potential crack sites and which may couple when any one propagates. We therefore use cohesive interface elements [3, 4] at all likely sites and let the structural response dictate the order of events. The only issue is the selection of the stress/displacement law for the interface element. Figure 7 shows the common bilinear law used in most codes and also an exponential law used by ourselves. Both laws model the linear growth to a maximum strength (like the ultimate tensile strength of a resin/matrix in mode I, or the shear strength in mode II) This avoids using the concept of an initial flaw like in linear elastic fracture mechanics. Thereafter the material degrades to zero stiffness such that the area under the curve is G_{IC} or G_{IIC}. We were cautious about the bilinear law having a negative stiffness [5] and so developed the monotonic exponential law. This has now been written into ABAQUS and has proved more accurate, but being a nonstandard routine, it is more expensive to run so we have not pursued it. Coupling between modes is enforced by the linear relationship

$$\frac{G_I}{G_{IC}} + \frac{G_{II}}{G_{IIC}} = 1 \qquad (3)$$

at which the interface fails even if $G_I < G_{IC}$ and $G_{II} < G_{IIC}$.

Secondly in-plane failure of matrix and fibres is modelled also using a form of damage mechanics to capture initiation before propagation. The details of such a strategy can be found in references [6, 7]. There may be coupling between intralaminar degradation and interlaminar failure, but this is left to the element models themselves to promote rather than using some empirical formula.

Validation of simulation

All the simulations presented in this paper have experimental tests also for validation. However even experimental results have uncertainty and variability, so it is desirable to test strategies on (relatively) simple benchmarks which have direct theoretical solutions. The first example is taken from [8, 9] and is an axisymmetric isotropic circular plate loaded by a normal pressure, and supported around the circular boundary at which a mid-plane crack is inserted. The plate has a uniform thickness which makes an axisymmetrical lay-up impossible so the real tests were conducted on a quasi-isotropic stacking sequence. If the through-thickness shear strength is used as a failure criterion then the pressure would need to increase as the crack propagates inwards since the shear decreases linearly from the edge to zero at the centre. A fracture mechanics prediction on the other hand decreases in the usual fashion from infinity for a zero flaw size. Figure 8 shows that the cohesive interface model switches correctly from the strength-based curve to the fracture-based curve at a crack length of 3 mm, and it agrees closely with the experimental results.

Another challenging test is that of a DCB with a starter crack at the loaded end but also a secondary crack inserted halfway along the beam as shown in Fig. 9. (see [10] for details) The test is a ‘hard’ one, that is imposed displacements, so when the crack propagates the reaction loads decrease. Figure 10 shows the history as predicted by the FE model. After the linear loading, the end crack follows the usual unloading path at constant energy release rate until the crack front approaches the secondary one, at which point there is a sudden decrease in the beam’s flexural stiffness and a dynamic drop from 40 N to near 25 N occurs. The beam is then reloaded at this reduced stiffness until both cracks propagate together at a net energy release rate, with the

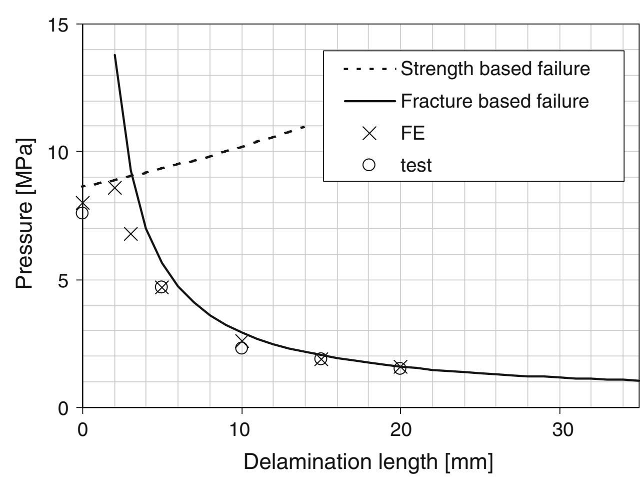


Fig. 8 Critical pressure against crack length for circular plate

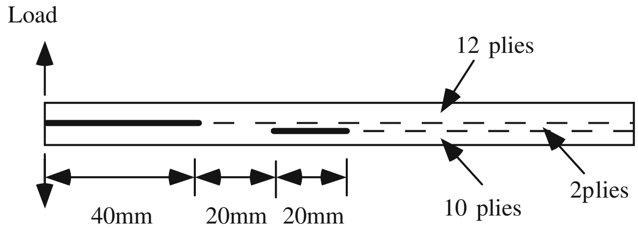


Fig. 9 DCB with a crack length 40 mm and a secondary crack of 20 mm inserted

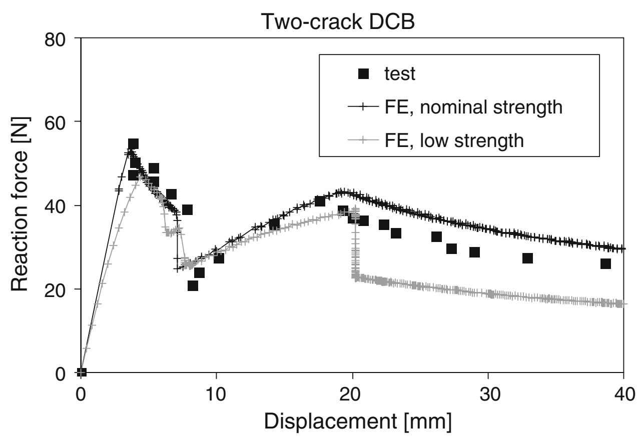


Fig. 10 Two crack DCB results using bilinear interfaces but one has artificial strength

secondary front leading. The bilinear interface elements have performed well, but also shown is the result for this interface having an artificially low strength and which suffers a sudden drop at 20 mm. This was published originally in the ABAQUS manual, where the artificially low yield stress was used to increase the length of the process zone (see Eq. 1) and enable a coarser mesh to be used. The dangers of such a ruse are clear.

BAESystems component tests

In 2003 BAESystems wished to find the strength of a new bonded stiffener/skin joint. The component test rig is

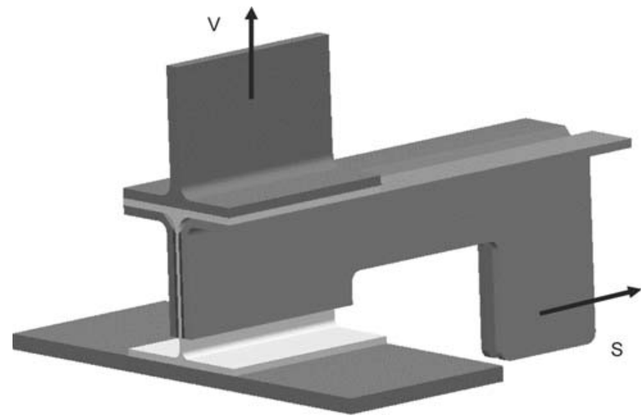


Fig. 11 Test rig for assessing stiffener/skin joint strength. (courtesy BAESystems)

shown in Fig. 11 for the two loading cases. It was realised that the strength depended crucially on the stress concentrations in the triangular region bounded by the two fillet radii and the skin: the so-called noodle. Four geometries were therefore selected: large and small radii, and thick and thin webs. Including hot and wet environments, some 200 tests were planned.

In 2004/5 a 2-D FE simulation was completed by positioning interface elements in the high stress zones revealed by the FE analysis. Further details can be found in [9]. In 2005 some 16 tests were performed for this tension loading case to failure. Figure 12 shows the good agreement between FE and experimental failure zones. It was predicted that all three zones delaminated almost simultaneously, and the experimental test also confirmed this unstable failure occurring within six frames of a 60,000 fps camera. Two interface thicknesses were modelled to vary the stiffness and in Fig. 13 the second figure shows little sensitivity to this effect. The first figure (for the strongest, large radius thick web.) shows the thinnest resin starts to fail earlier and seems to explain the larger variation in the experimental results.

The shear loading case present a much greater challenge since the model now has to be fully three dimensional, and shown in Fig. 14 consists of 130,000 elements! The

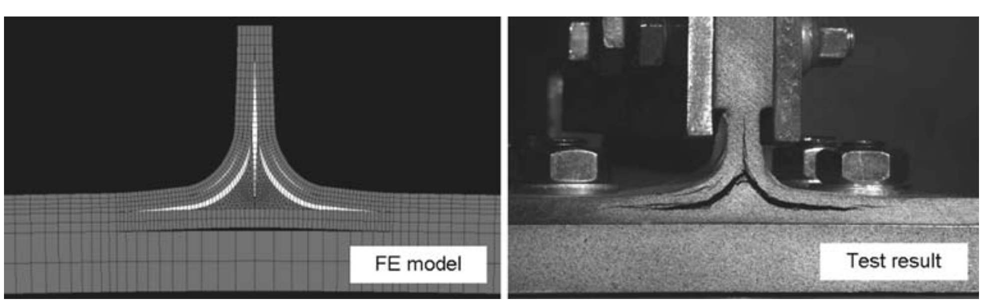


Fig. 12 Final delamination zones after failure, prediction, and test

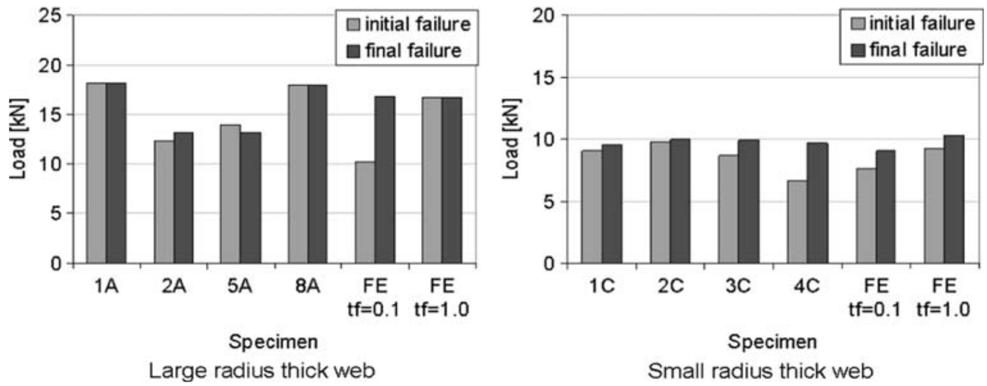


Fig. 13 Failure loads and FE predictions for stiffener under tension loading

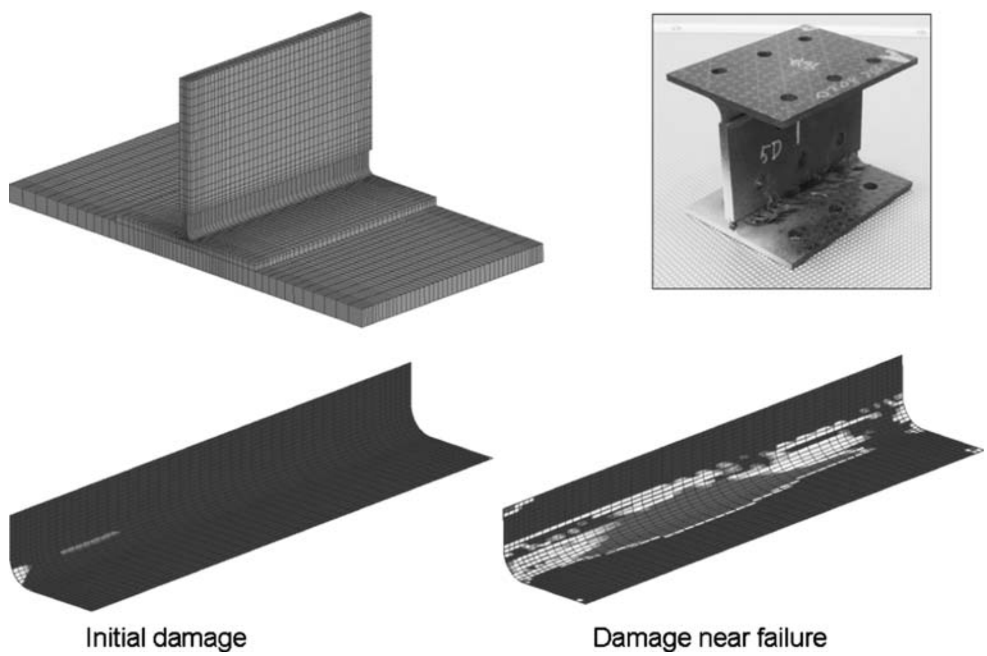


Fig. 14 FE model and consequent damage maps

damage initiated at one end and proceeded to spread gradually along the joint, but the calculation was terminated after 14 CPU hours when the stiffness had fallen to a negligible value, with automatically selected load increments also falling to very small amounts. In spite of this, the agreement with test failure values was good, except for the two stronger configurations for which the steel loading arm yielded!

This poses a dilemma, since a computing time of 14 CPU hours is not acceptable for a design tool. The explanation is that ABAQUS Standard has to solve all equations at each load step in spite of the fact that the advancing damage front is affecting only a small number of elements. An adaptive moving local region is not possible in this code at present. Our solution has therefore been to turn to ABAQUS Explicit which avoids solving equations. The necessarily small time step can be increased for static problems by using selective mass scaling proportional to element stiffness. There is an added bonus that this code version can be parallelised and run on multiprocessors. This strategy is easy to use and the system organises the domain partitioning for the user.

Finally it should be said that we have earlier in our in-house code used an explicit solver with zero mass, that is posing a purely viscous problem with no critical time step. Further, by putting the damping matrix $C = K$, the entire structure has the same exponential decay rate. The down side to this ruse is that K has to be inverted at a time when the tangent stiffness matrix is changing due to damage. To avoid this we can lump $C = I$, and accept a multitude of decay rates. These strategies are not yet available commercially.

Conclusions

In spite of the fact that carbon-fibre composite structures are so vulnerable to even very local stress concentrations, it seems that simulating a component test to failure, including the death throes, is possible for a structure with complex failure mechanisms, and in an acceptable computing time for a realistic design tool. It does seem that an explicit solver is to be preferred when the damage zone propagates in a stable fashion before failure. A large number of finite elements may be needed to cope with a small process zone, so advantage will be taken of running an explicit solver on parallel machines. Industry, both military and civil, will be undertaking many and varied simulations to lend credibility to this strategy.

Acknowledgements The authors would like to thank BAESystems who have supported this analytical and experimental work for many years.

References

1. Dugdale DS (1960) J Mech Phys Solids 8:100
2. Hinton M, Soden P, Kaddour AS (2004) Failure criteria in fibre-reinforced polymer composites. Elsevier, Amsterdam
3. Schellekens JCJ, de Borst R (1993) Int J Solids Struct 30(9):1239
4. Corigliano A (1993) Int J Solids Struct 30(20):2779
5. Mi Y, Crisfield MA, Hellweg H-B, Davies GAO (1998) J Compos Mater 32(14):1246
6. Davila CG, Camanho PP (2003) Failure criteria for FRP laminates in plane stress. NASA TM 212663
7. Maimi P, Camanho PP, Mayugo JA, Davila CG (2006) A thermodynamically consistent damage model for advanced composites. NASA TM 214282
8. Davies GAO (2002) Benchmarks for composite delamination. NAFEMS report R0084
9. Davies GAO, Hitchings D, Ankersen J (2005) Compos Sci Technol 66:846
10. Robinson P, Besant T, Hitchings D (1999) Delamination growth prediction using a finite element approach. 2nd ESISTC4 conference on polymers and composites, Les Diablerests, Switzerland