INITIATION AND EVOLUTION OF CRACKS IN A CHOPPED GLASS-REINFORCED COMPOSITE UNDER BIAXIAL TESTING BY MEANS OF XFEM

M. C. Serna Moreno[a*], J. L. Curiel Sosa[b], J. Navarro Zafra[b]

[a]Instituto de Investigaciones Energéticas y Aplicaciones Industriales, Escuela Técnica Superior de Ingenieros Industriales de Ciudad Real, Departamento de Mecánica Aplicada e Ingeniería de Proyectos, University of Castilla-La Mancha, Avenida Camilo José Cela s/n, 13071 Ciudad Real, Spain.
[b]Composite Systems Innovation Centre, Department of Mechanical Engineering, University of Sheffield, Sir Frederick Mappin Building, Mappin Street, S1 3JD Sheffield, United Kingdom.
**e-mail: mariacarmen.serna@uclm.es*

Keywords: XFEM, Biaxial testing, Chopped glass-reinforced composite, crack initiation and evolution.

Abstract

Modelling the initiation and propagation of cracks in composites using the Finite Element Method (FEM) has always been problematic. However the so-called XFEM (eXtended Finite Element Method), first introduced by Belytschko and Black (1999), has overcome the principal difficulties for modelling discontinuities in FEM: it reduces the necessity of remeshing during the crack progress and it can be used without including in the initial geometry a pre-crack. This new technique provides an effective engineering approach which here will be applied to simulate the initiation and growth of cracks in a chopped glass-reinforced composite submitted to biaxial loading. The test consists on applying quasistatic in-plane loads in the arms of a cruciform specimen, in which the biaxial stress state will occur in its central part.

1. Introduction

In any sector of the industry it is crucial to study the ultimate limit state of the resistant structures. For example, it becomes particularly important in wind power generators (Figure 1), in which one of the critical points of their structure are the blades. These are mostly made of materials composed by polymeric resin and reinforced with glass fibres, which are usually subjected to complex loading conditions. This makes fundamental for their correct use to determine the processes that lead to the failure of these materials under loading cases different from the uniaxial.

Figure 1. Source: http://www.theguardian.com/technology/2008/sep/04/energy.engineering. (The Guardian, "Spinning to destruction", 4 September 2008.

Multiaxial loading conditions are usually difficult to implement and successfully reproduce in the laboratory. In previous works [1,2] biaxial stress states have been achieved in the central area of cruciform specimens submitted to tensile loading on their arms (Figure 2). From the experimental results it has been obtained the fracture curve of a material composed by 80 % of polyester resin and 20% of glass-fibres randomly distributed, when it is subjected to tensile loads applied simultaneously in two perpendicular directions. This work pursues to test whether these experimental results agree with the failure predictions obtained by means of XFEM (eXtended Finite Element Method) [3,4].

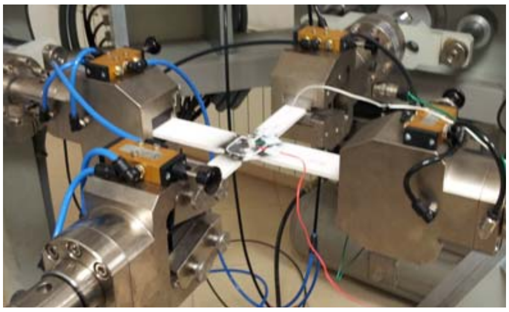

Figure 2. Experimental set-up [2]: the cruciform specimen, the application of the biaxial loading by means of electromechanical actuators and the strain measurement with strain gages.

From the numerical point of view, it has been always problematic to model the processes of damage initiation and evolution in different materials. However, the XFEM has managed to overcome the main problems that arise in the classical form of modelling. With XFEM it is not necessary to include a pre-crack in the initial geometry and it alleviates the necessity of performing an adaptive remeshing to conform the crack during the simulation.

Although the XFEM is a relatively new method of analysis (1999), it has been already implemented in several commercial software of finite elements. This work aims to evaluate whether the XFEM of ABAQUS [5] is suitable for modelling the process of crack initiation and progression under biaxial loading states and to understand the possibilities that it offers for a general analysis [6-12]. Therefore, we have tried to analyze numerically the onset and the evolution of cracks that lead to the fracture of the cruciform specimens tested experimentally [1,2]. The intention is to compare the results generated using the ABAQUS XFEM with those obtained experimentally.

2. Numerical framework

In the Finite Element Method (FEM) the displacements produced in the domain of an element are approximated from the nodal values using the element shape functions. The method is based on the concept denominated "partition of unity", which means that the sum of the values that the shape functions take in a certain degree of freedom has to be equal to one. The XFEM is also based on the notion of "partition of unity" to define the approximation of the displacements produced on an element. In this case the approximation must include the displacement discontinuity due to the presence of the crack and the asymptotic behaviour of the displacement in the area near the crack tip (due to the stress and strain concentrations). In crack propagation problems as the one which competes us, the displacement approximation performed on an element containing the crack, called enriched element, must include a term that includes the so-called "enrichment function" or "jump function". This term forces that the displacement discontinuity occurs in the elements containing the crack, maintaining the classical approach of the displacements (FEM) in the other elements of the mesh.

For simplicity reasons related with the geometry and the runtime, in this study we have chosen to work with plane models. In XFEM it is only possible to work on the plane with continuous solid elements with reduced integration, so that, it has been chosen to use the element CPS4R. This is a continuous solid element of four nodes, working on plane stress, which considers a single integration point located in the centre of the element for the numerical integration of the stiffness matrix and the load vector. The geometry and dimensions of the three specimens studied (geometries A, B and C) are defined in [1]. But in this work, the different thicknesses of each region are specified giving the element perpendicular dimension to the working plane.

It has been considered a linear elastic behaviour of the material followed by a process of damage initiation and evolution. The material is quasi-isotropic and its linear elastic behaviour is defined by the elastic modulus $E = 6.5$ GPa and the Poisson coefficient $\nu = 0.37$. The damage initiation criterion is given in terms of the stresses. Then, the damage occurs on an element when the maximum principal stress reaches a critical value (within a tolerance defined by the user), that in this study it has been chosen equal to the yielding strength of the material $\sigma_Y = 90$ MPa. The criterion considers that those enriched elements that are subjected to a state of pure compression in the first principal direction will not suffer damage. After the damage initiation it is defined a linear softening behaviour of the material, which means that the damage evolution follows a law that decreases linearly until the fracture. The program requests to specify the effective displacement which is produced at the moment of the fracture, measured from the moment at which the damage begins.

To perform the numerical integration of the equations of equilibrium an implicit scheme (ABAQUS/Standard) has been used, in which an iterative method is utilised to find the solution. Then, a time step too large may prevent us from reaching the solution if the initial state is too far from the equilibrium state and, so that, convergence problems may appear. In this work we approximate the characteristic time of the system from the propagation velocity of the stress waves in a material and the smallest dimension of the elements in the model.

3. Results

Non-linear static analyses have been developed and a total time equal to 1 is assigned to each analysis. In this case the time scale is not real, it is a measure used to indicate the total application of the load, describing then the time increments the loading history. The analyses have been performed under the assumption of large displacements, considering an applied stress on the arms of the cruciform specimen equal to 100 MPa in a total time equal to 1. It is worth to highline that, among the three geometries studied (A, B and C), in geometry A there have been convergence problems. These convergence problems appeared at the same level of load that is applied in the experimental tests when the crack propagates almost instantaneously till the total fracture. The convergence problems were solved by reducing the time step, using a time step closer to the real time of the crack propagation in this last phase of growth.

All the analyses have reproduced the location of the crack initiation, the direction of the crack evolution and the load which provokes the total failure of the specimen. For example, in geometry A submitted to the same level of load on both arms of the specimens, the crack emerges in the external edge of the intersection between the two arms of the specimen. Then the crack growths with an angle approximately equal to 45º till a torn rupture is produced almost instantaneously at a time step equal to 0.56, which is equivalent to an applied stress of 56 MPa. This applied stress is approximately equal to the real level of the imposed load which produces the total fracture of the specimen.

4. Conclusions

The numerical analyses by means of XFEM have reproduced the experimental results without defining a pre-crack in the initial geometry or remeshing during the analysis. Therefore, the XFEM is able to predict the crack initiation and evolution, as well as the level of load that produces the total failure of the cruciform specimen under biaxial loading. So, the XFEM provides an effective and attractive engineering approach to simulate the initiation and growth of multiple cracks in solids under general loading conditions.

It is relevant to mentioned that this work has involved a deep study of the XFEM numerical model. It has been necessary to identify and determine the significant parameters from the experimental tests in order to implement them in the numerical code and to solve several convergence problems during the simulations.

Acknowledgements

This work has been financially supported by the University of Castilla-La Mancha of Spain under the grant CYTEMA.

References

[1] M. C. Serna Moreno and J. J. López Cela. Failure envelope under biaxial tensile loading for chopped glass-reinforced polyester composites, *Compos Sci Technol*, 72:91-96, 2011.
[2] M. C. Serna Moreno, J. L. Martínez Vicente and J. J. López Cela. Failure strain and stress fields of a chopped glass-reinforced polyester under biaxial loading, *Compos Struct*, 103:91-96, 2013.
[3] T. Belytschko and T. Black. Elastic crack growth in finite elements with minimal remeshing, *Int J Numer Meth Engng*, 45:601-620, 1999.
[4] N. Moës, J. Dolbow and T. Belytschko. A finite element method for crack growth without remeshing, *International Journal for Numerical Methods in Engineering*, 46:131-150, 1999.
[5] Abaqus 6.12 Online Documentation. Dassault Systmes, Providence, Rhode Island, 2012.
[6] X. P. Xu and A. Needleman. Numerical simulations of fast crack growth in brittle solids, *Journal of the Mechanics and Physics of Solids*, 42(9):1397-1434, 1994.

[7] S. Rudraraju, A. Salvi, K. Garikipati and A. M. Waas. Experimental observations and numerical simulations of curved crack propagation in laminated fiber composites, *Composites Science and Technology*, 72:1064-1074, 2012.

[8] A. Combescure, A. Gravouil, D. Grégoire and J. Réthoré. *Journal of Computer Methods in Applied Mechanics and Engineering*, 197:309-318, 2008.

[9] S. Natarajana, D. R. Mahapatrab and S. P. A. Bordasc. X-FEM a good candidate for energy conservation in simulation, *International Journal for Numerical Methods in Engineering*, 83(3):269-294, 2010.

[10] E. Giner, N. Sukumar, J. E. Tarancón and F. J. Fuenmayor. An Abaqus implementation of the extended finite element method, *Engineering Fracture Mechanics*, 76:347-368, 2009.

[11] R. Huang, N. Sukumar and J. H. Prévost. Modeling quasi-static crack growth with the extended finite element method Part II: Numerical applications, *International Journal of Solids and Structures*, 40:7539-7552, 2003.

[12] J.L. Curiel Sosa and N. Karapurath. Delamination modelling of GLARE using the extended finite element method, *Composites Science and Technology*, 72:788-791, 2012.