MODELLING OF DEBOND ONSET AND GROWTH IN A GROUP OF FIBRES UNDER TRANSVERSE LOADS USING ABAQUS USER SUBROUTINE UMAT

D. Castillo[a*], L. Távara[a], V. Mantič [a]

[a]*Grupo de Elasticidad y Resistencia de Materiales, Escuela Técnica Superior de Ingeniería, Universidad de Sevilla, Camino de los Descubrimientos s/n, 41092 Sevilla, Spain*
**dvdcastillog@gmail.com, ltavara@us.es, mantic@us.es*

Keywords: LEBIM, debond onset, FEM, UMAT

Abstract

A FEM study of debond onset and growth at fibre- matrix interfaces in a single fibre and in a group of ten fibres subjected to remote transverse load is presented. The debond is modelled using the Linear Elastic Brittle Interface Model (LEBIM) which is implemented in the commercial FEM code Abaqus using a user subroutine UMAT. In both cases studied, the external dimension of the matrix is much larger than the fibre radius. The purpose of this paper is to predict the critical loads that produce crack onset and to compare the obtained results with previous BEM analyses.

1. Introduction

In this paper debonds produced along the interfaces of a glass fibre/epoxy composite under remote transverse tension are analysed using the LEBIM assuming plane strain conditions. A detailed study and several applications of the LEBIM in composites can be found in [1]. The deboning problem of an elastic circular inclusion embedded in an elastic matrix was studied using the LEBIM and BEM in [1-3]. The debonding problem in a group of ten fibres is studied in [4]. Many authors studied different debonding problems for a single fibre and multiple fibres in a unit cell by using FEM under assumptions of a cohesive behaviour of the interface, see e.g. references [5- 7]

The present paper is organized as follows. First, a brief review of the LEBIM is presented. After a brief description of the two models considered, the algorithm implementation and the strategy used are explained in detail. Then the numerical results obtained are showed and compared with the results obtained by previous BEM analyses. Finally, a future work and applicability of the methodology developed to more realistic studies of composites are discussed.

2. Linear elastic brittle interface model (LEBIM)

In this section, the LEBIM and its interface failure criterion is briefly explained. For further information, see [1,2]. The interface is modelled as a continuous spring distribution along the fibre-matrix interface. The behaviour of this interface is described by a simple linear elastic brittle law, written at an interface point x as:

Linear elastic interface $\begin{cases} \sigma(x)=k_n\delta_n(x) \\ \tau(x)=k_t\delta_t(x) \end{cases} \quad t(x)< t_c(\psi_\sigma(x))$

$$\text{Broken interface} \quad \begin{cases} \sigma(x)= \begin{cases} 0 & \delta_n(x) > 0 \\ k_n\delta_n(x) & \delta_n(x) \leq 0 \end{cases} \\ \tau(x)=0 \end{cases} \tag{1}$$

where $\sigma(x)$ and $\tau(x)$ are, respectively, the normal and shear stresses along the interface in a very thin elastic layer. The normal and tangential relative displacements between opposite interface points are represented by $\delta_n(x)$ and $\delta_t(x)$. k_n and k_t denote the normal and tangential stiffnesses of the spring distribution. The traction modulus t is defined as $t(x)= (\sigma^2(x)+ \tau^2(x))^{1/2}$.

The failure criterion is expressed as $t(x)<t_c(\psi_\sigma(x))$, where $\psi_\sigma(x)$ is the fracture-mode-mixity angle at an interface point x. This angle is defined by $\tan \psi_\sigma(x)= \tau(x)/ \sigma(x)$. Therefore, the critical traction modulus, in general, may depend on the position along the interface. It should be mentioned that this criterion is motivated by an energetic criterion, although it is finally expressed as a traction criterion. An extensive explanation of the deduction of this model can be found in [1-3, 8]

3. Model description

Both models presented and studied in this paper are the same as the ones which appear in references [2, 4], so the obtained results can be compared. The models considered a single or a group of fibres inside a large matrix. The matrix used in the models is a 1 mm side square, whereas the fibres have a radius a=7.5 µm. A plane strain state is assumed. Both fibres and matrix are considered as linear elastic materials. Two different coordinate systems are used (x,y,z) being the Cartesian system and (r,θ,z) being the cylindrical system, the z-axis is the longitudinal axis, and the x-axis is parallel to the direction of the transverse load. The matrix sides parallel to the y-axis are subjected to a uniform remote tension $\sigma_x^\infty>0$

3.1. Single fibre

The first problem considered is a single fibre embedded in an infinite matrix (unit cell dimensions much larger than the fibre radius). In Figure 1, the coordinate systems used are shown.

(a) (b)

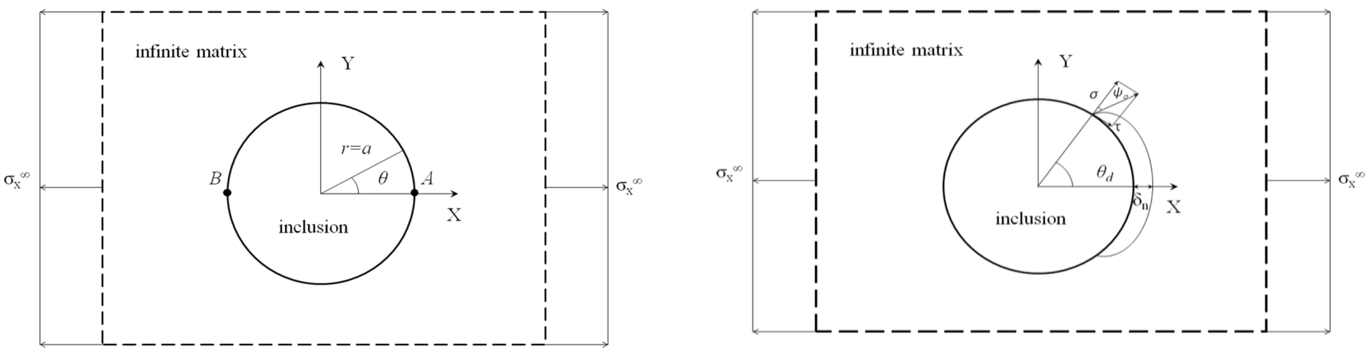

Figure 1. Single fibre problem under remote tension (a) without and (b) with a partial debond

To make the problem more realistic, from an experimental point of view, only debond at one side is considered. However, as can be noticed the problem described before is a symmetric problem so the solution obtained may be symmetric too, in others words, the debond may occur simultaneously at two points of the interface. To obtain a debond at one side only, a portion of the interface (of 45º angle) in the neighbourhood of point B, see Figures 1(a) and 4, is considered to be more resistant than the rest of the interface. Thus, once the crack has started at one side, it will continue growing along this side only.

3.2. Group of ten fibers

A bundle of ten infinitely long cylindrical inclusions is considered. The positions of the fibres correspond to a portion of an actual glass fibre composite micrograph [4]. A uniform remote load in direction x is supposed too.

4. FEM solution strategy

In this section the present LEBIM implementation in the FEM code Abaqus [10] is explained. After that, a description of the problem instabilities and how the problem is solved is presented.

4.1. LEBIM implementation

A User Material (UMAT) subroutine in Abaqus is used to implement the LEBIM described in Section 2. A brief explanation is shown in Figure 2.

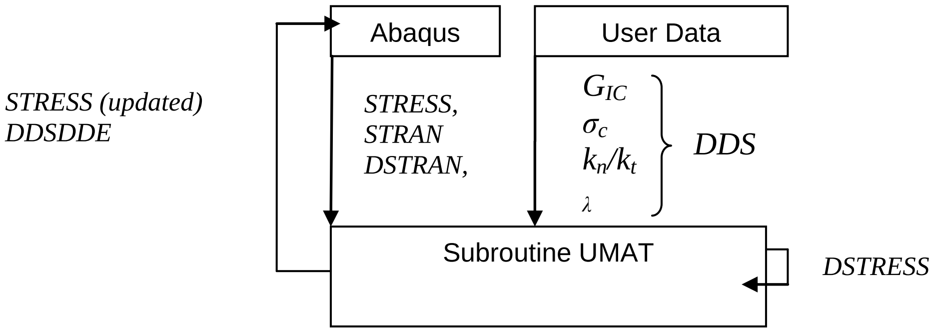

Figure 2. Scheme of LEBIM - UMAT

The interface behaviour is defined in the subroutine defining the stiffness matrix (DDS), in our case, this matrix is a 3x3 diagonal matrix. To obtain the stiffness of the interface, the following variables G_{IC}, σ_c and k_n/k_t are needed. Stress, strain and increment of strain are input data into the UMAT subroutine obtained from Abaqus. We are able to actualize these variables in the subroutine. In Figure 2, DDS is the stiffness matrix, STRESS is the stress pseudo-vector, STRAN is the strain vector, DDSDDE is the updated stiffness matrix, and DSTRAN is the updated pseudo-strain vector.

This subroutine is executed for each time increment, and for every integration point of all elements. It should be noted that, none variable could be passed from one increment to other except the one stored in STATEV variable, defined for every integration point. Thus, STATEV variable is going to be used as a damage variable to know whether some of the integrations points have reached the failure criterion.

The CPE4 element of Abaqus is used, this elements has 4-integration points. The extrapolation to determine the variables at the element nodes may lead to wrong results in some cases. For example, it could happen that there are some integration points broken and others without damage within an element. In this case, extrapolation of the results to the nodes may cause that some nodes will be in compression and other will have stresses a bit higher than the critical one. To sum up, we can not be sure that the results at nodes are correct (due to the extrapolation) although we do know that the results at integration points are correct. That is why, the results in integration points should be used in the post processing.

4.2. Instabilities and solution methods

In some papers [1-6], it can be observed that a snap trough instability is produced in both studied models. Some of these authors have been able to model this behaviour by means of BEM and a sequentially linear analysis [1-4]. However, in FEM, authors solved this problem using an automatic stabilization, in others words using a fictitious damping factor [7, 9]. In this paper the single fibre model is solved using first the Newton-Raphson method and second with an automatic stabilization. The model of ten fibres is solved using only the automatic stabilization, because of the difficulties that Newton-Raphson has to converge in this problem. In the following some details are presented.

Single fibre problem. To obtain the convergence of the problem using Newton-Raphson, the "general solution control" of Abaqus is modified. One of the modifications is to control how many iterations Abaqus makes for every increment, for this problem at least 12 iterations should be done for every increment. This problem is also solved with the automatic stabilization, in this case Abaqus calculate the damping factor in every increment, because a $2.5\text{x}10^{-4}$ fraction of dissipated energy fraction is imposed. With this strategy, the interface elements can be broken one by one, but the solution during the snap through is not in equilibrium.

The model of ten fibres is solved using the automatic stabilization, but this time the damping factor is imposed. The value of the damping factor used is $3.5\text{x}10^{-8}$. This value was obtained by trial and error because we can compare the FEM solution with the solution previously obtained by BEM [4].

5. Results and comparison with a BEM analysis

A glass fibre and epoxy matrix system is used in both problems. The elastic properties of these materials are:

Material	Poisson's ratio	Young's modulus
Matrix	$\upsilon_m = 0.33$	$E_m = 2.79$GPa
Fibre	$\upsilon_f = 0.22$	$E_f = 70.8$GPa

Table1. Elastic constants of isotropic materials

In addition, the LEBIM needs the input of four independent variables: the critical tension in mode I (σ_c), the fibre-matrix interface fracture toughness in mode I (G_{Ic}), the fracture mode sensitivity parameter (λ) and k_n/k_t ratio. The first two values are: σ_c=90 MPa and G_{Ic} =2Jm^{-2}. These values are chosen because they are in the range of values found in the literature [11, 12] and simulate a brittle interface behaviour [4] making the hypothesis of the LEBIM to appropriately represent a possible real composite material behaviour. The values $k_n/k_t = 4$ and $\lambda = 0.25$ are used. The same values were used in the BEM analysis.

5.1. Single fibre results

The FEM model represents a cylindrical inclusion with a radius a=7.5 μm inside a relatively large square matrix with a 1 mm side, and an interface thickness of 0.01 μm. The interface has 360 elements, the fibre has 3905 elements with small elements near to the interface, and the matrix has 15190 elements, which are small near the interface too. The mesh used is shown in figure 3.

(a) (b)

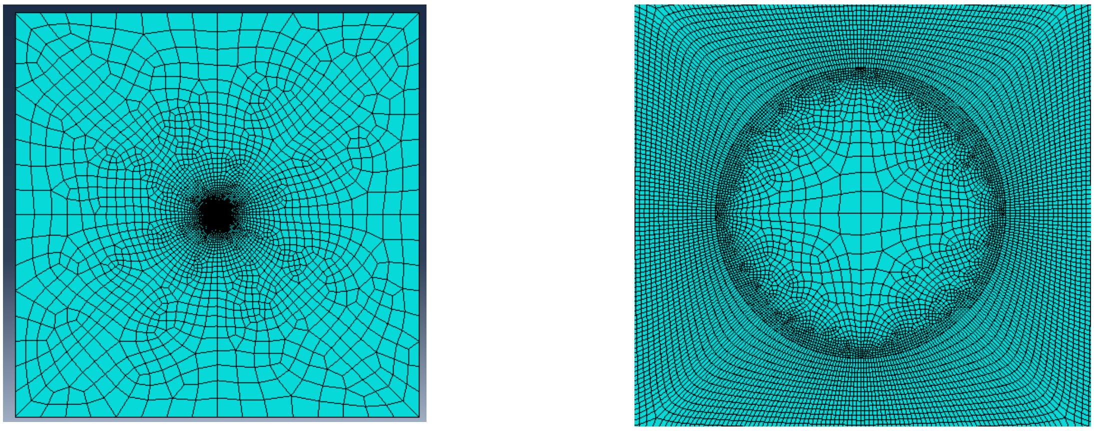

Figure 3. Mesh of the whole model (a) and a detail of fibre's mesh (b)

As mentioned above, in this study a little portion of the interface (of 45º angle) is considered to have a greater critical tension in mode I (σ_c=270 MPa, 3 times the original), to impose debonding at one side only. In figure 4, the interface portion with higher strength is shown.

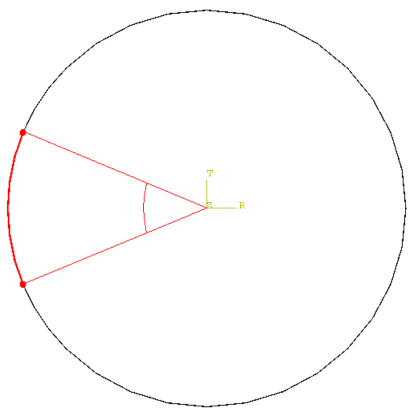

Figure 4. Portion of the interface which is considered to have a higher strength

In Figure 5 the applied remote stress, σ_x^{∞}, is plotted as function of the normal relative displacement (opening) δ_n, evaluated at point A (a, 0º) as defined in Figure 1.
As it can be seen in Figure 5 there is an instability, whose complete solution can be obtained by using BEM and the algorithm detailed in reference [2]. But using the present UMAT subroutine in FEM with Newton-Raphson or automatic stabilization the whole equilibrium

path of instability can not be reproduced. For more details about the instability produced in this model, see references [1-3].

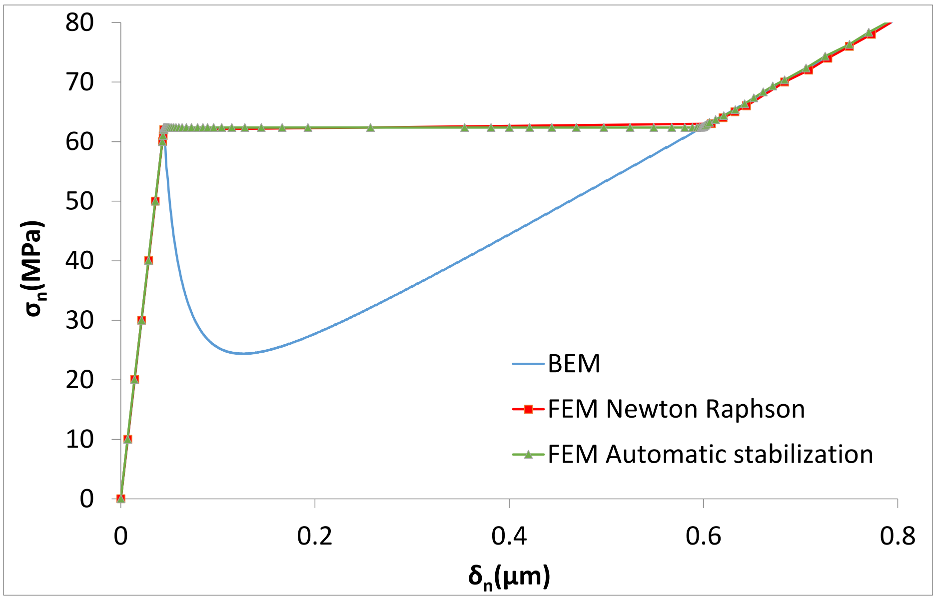

Figure 5. Remote stress as function of normal relative displacement

5.2. Group of ten fibres results

In this 2D model, the same dimensions are used for matrix, fibres and interfaces. However, the difference between this model and the previous model is that in this case a portion of the interface with different properties is not necessary, as the proximity between fibres produces a stress concentration effect, so one side debonds may appear. Meshes of 360 elements for the fibre-matrix interface and 3905 for each fibre are again used, whereas the mesh of the matrix has 49788 elements. Similar to the single fibre case, the mesh is more refined around the interface.

(a) (b)

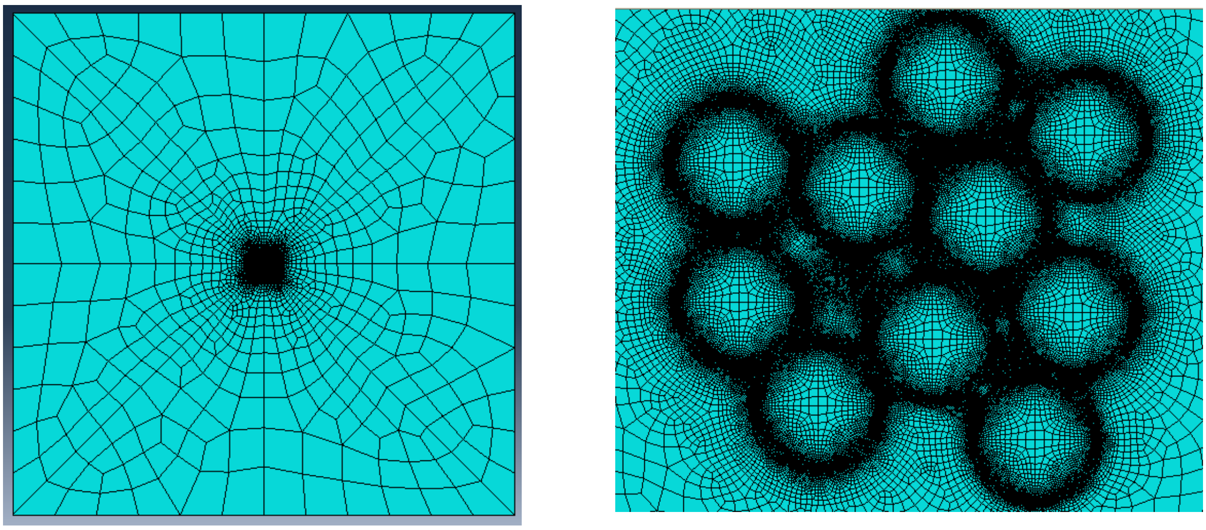

Figure 6. (a) mesh of the whole ten-fibres model, and (b) detail of the mesh for fibres and neighbourhood.

The results obtained in [4] for the model of ten fibres show that, a sequence of local unstable growths of debonds occurs. Specifically a series of picks and valleys occurs in the remote-stress plot, each of them corresponding to a debond onset and subsequent unstable growth at one of the different fibres. Using automatic stabilization in Abaqus, this unstable behaviour can not be reached, but only a unique snap-through solution is obtained. A 3.5×10^{-8} value as damping factor is used. In Figure 7, the crack path is obtained by using the present FEM procedure and by BEM.

The value of the remote tension that causes these debonds is 45.13 MPa for the FEM analysis while in the BEM analysis a value of 45.27 MPa was obtained.

(a) (b)

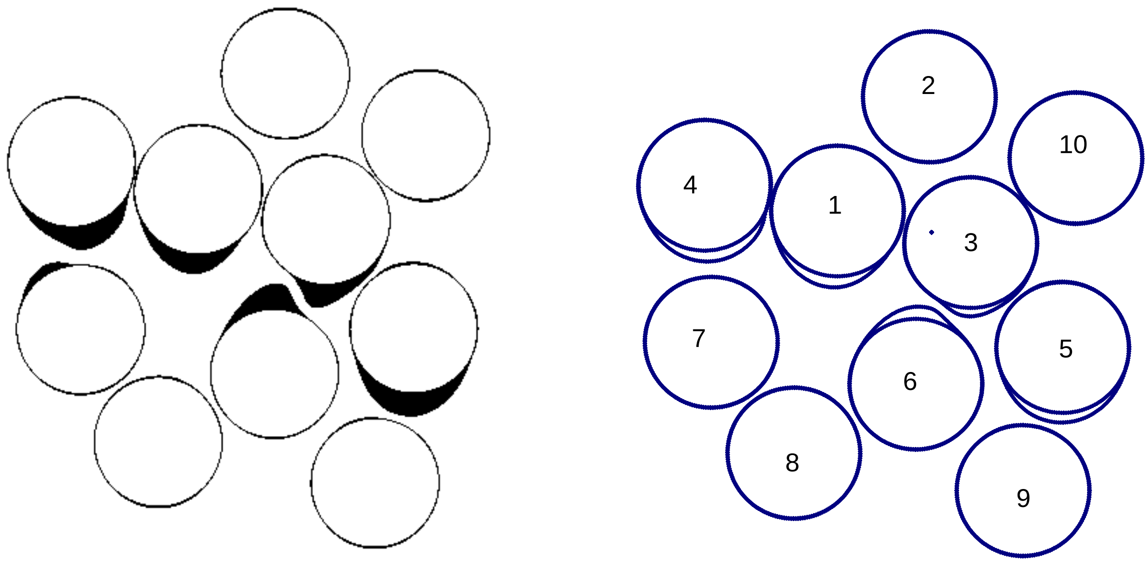

Figure 7. Crack path obtained in (a) the present FEM analysis and (b) BEM analysis in [4]

6. Conclusions

In this paper the implementation of the LEBIM to model the interface behaviour in Abaqus using a UMAT subroutine is presented. The most important result of the present work is the potential for future developments and applications obtained by including the LEBIM in Abaqus. It can be observed that the results obtained are in good agreement with those obtained in a previous BEM analysis, proving the correct implementation of the model. Some advantages of the use of a FEM commercial code is that problems with more complex geometries could be modelled in an easy way.

It should also be stressed that this paper presents preliminary results only. Future work includes a model with a greater number of fibres, around 100. In addition, a deeper analysis of the solving process will be done, including the programing of an algorithm capable to capture the instabilities that may appear.

Acknowledgements

The work was supported by the Junta de Andalucía and European Social Fund (Projects of Excellence TEP-1207, TEP-2045, TEP-4051, P12-TEP-1050), the Spanish Ministry of Education and Science (Projects TRA2006-08077 and MAT2009-14022) and Spanish Ministry of Economy and Competitiveness (Projects MAT2012-37387 and DPI2012-37187).

References

[1] L. Távara. Damage initiation and propagation in composite materials. Boundary element analysis using weak interface and cohesive zone models. PhD Thesis. Universidad de Sevilla: Sevilla 2010.

[2] L. Távara, V. Mantič, E. Graciani, F. París. BEM analysis of crack onset and propagation along fibre-matrix interface under transverse tension using a linear elastic-brittle interface model. *Engineering Analysis with Boundary Elements*, 35: 207-202, 2011.

[3] V. Mantič, L. Távara, A. Blázquez, E. Graciani, and F. París. Crack onset and growth at fibre-matrix interface under transverse tension using a liner elastic-brittle interface model. *ArXiv preprint. arXiv:1311.4596, 2013.*

[4] L. Távara, V. Mantič, E. Graciani, F. París. BEM modelling of interface cracks in a group of fibres under biaxial transverse loads. *Advances in Boundary Element Techniques XIV*. 311-316, 2013.

[5] A. Caporale, F. Luciano, E. Sacco. Micromechanical analysis of interfacial debonding in unidirectional fiber-reinforced composites. *Computers and Structures*, 84:2200-2211, 2006.

[6] V.I. Kushch, S.V. Shmegea, L. Mishnaevsky Jr. Explicit modeling the progressive interface damage in fibrous composite: Analytical vs. numerical approach. *Composite Scienci and Technology*, 71:989-997, 2011.

[7] M. Romanowicz. Progressive failure analysis of unidirectional fiber-reinforced polymers with inhomogeneous interphase and randomly distribute fibers under transverse tensile loading. *Composite Part A:appalied science and manufacturing*, 41:1829-1838, 2010.

[8] L. Távara, V. Mantič, E. Graciani, J. Cañas, and F. París. Analysis of a crack in a thin adhesive layer between orthotropic materials. An application to composite interlaminar fracture toughness test. *Computer Modeling in Engineering and Sciences*, 58:247-270, 2010.

[9] R.T. Tenchev, B.G. Falzon. A pseudo-transient solution strategy for the analysis of delaminating by means of interface elements. *Finite Elements in Analysis and Design*, 43:698-708, 2006.

[10] Abaqus User Subroutines Reference Manual. V 6.12.

[11] V. Mantič. Interface crack onset at a circular cylindrical inclusion under a remote transverse tension. Application of a coupled stress and energy criterion. *International Journal of Solids and Structures*, 28A:1287-1304, 2009.

[12] H. Zhang, M.L. Ericson, J. Varna, and L.A. Berglund. Transverse single-fiber test for interfacial debonding in composites: 1. Experimental observations. *Composites Part A: Applied Science and Manufacturing*, 28A: 309-315, 1997