

A COMPARATIVE NUMERICAL STUDY AIMING TO REDUCE COMPUTATION COST FOR MODE-I DELAMINATION SIMULATIONS

S. AhmadvashAghbash¹, M. Engül¹, F. E. Öz¹, R. Amali², and N. Ersoy^{1,2}

¹ Department of Mechanical Engineering, Boğaziçi University, Bebek, 34342, Istanbul, TURKEY
Emails: sina.ahmadvashaghbash@boun.edu.tr, mehmet.engul@boun.edu.tr, fatih.oz@boun.edu.tr

² Department of Engineering, Design, and Mathematics, University of West of England, Bristol,
UNITED KINGDOM

Emails: ramin2.amali@uwe.ac.uk, nuri.ersoy@boun.edu.tr

Keywords: Double Cantilever Beam, Cohesive Zone Model, Mode I Fracture Toughness, Interfacial Strength

Abstract

This study presents a procedure which aims to find a solution that allows the use of coarser mesh in modelling the Mode I delamination behaviour of AS4/8552 Carbon-Epoxy laminates. This approach is based on an artificial increase of the cohesive zone length by lowering the interfacial strength. In this paper, Double Cantilever Beam (DCB) tests are performed experimentally to obtain the load-displacement curve and to evaluate the mode I interlaminar fracture toughness value of AS4/8552. Moreover, 2D and 3D numerical models are applied to simulate DCB test with commercial Finite Element Analysis (FEA) software ABAQUS. A parametric study is carried out for both models to investigate the effect of various parameters such as mesh size, interfacial strength and interface stiffness values. Results shows that the propagation of delamination can be accurately predicted with coarser meshes by optimizing the parameters used in Cohesive Zone Modelling.

1. Introduction

The urge to reduce the design costs of composite structures relying on experimental testing has encouraged researchers to develop advanced computational tools to simulate the progressive failure of fibre-reinforced composite materials. Among the different failure mechanisms that occur in laminated composites, delamination is one of the predominant ones. However, the simulation of progressive delamination contains numerical difficulties related to the proper definition of the interlaminar properties and the requirement of extremely refined mesh which leads to increase in run-time. Therefore, developing a numerical approach for delamination growth by using coarser mesh is quite substantial.

Cohesive Zone Model (CZM) is one of the common approaches that is performed to simulate delamination by using decohesion elements [1-6]. Decohesion elements predict the onset of the softening process at the interface between plies by strength-based approach and fracture mechanics to predict delamination propagation. Compared to alternative methods such as Virtual Crack Closure Technique (VCCT) [7], CZM provides the ability to have several crack paths without predefined crack propagation directions.

Turon et. al. [8] proposed an approach based on an artificial increase of the cohesive zone length by reducing the interfacial strength while keeping the fracture toughness constant. Meanwhile, the number of elements is kept constant in the cohesive zone, where the increase of cohesive zone length leads to

an increase of mesh size. In this paper, the procedure is applied to find a solution that allows the use of coarser meshes in the delamination simulations for AS4/8552 carbon-epoxy laminates.

2. Overview of the Cohesive Zone Model (CZM)

The primary study on CZM backs to the works of Barenblatt [9] and Dugdale [10] on the fracture behaviour around the crack tip in brittle materials and steel. Hillerborg et al. [11] combined fracture mechanics with FEA to obtain more realistic results on crack formation and evolution. The CZM considers a cohesive zone (process zone) ahead of the crack tip to count for crack initiation and extension by using sets of constitutive equations such as traction-separation laws. The traction-separation laws associate the opposing tractions of the cohesive zone to the opening displacements. Depending on the crack opening mode (Mode I, II or III), the traction-separation laws are defined based on initial stiffness, maximum traction, interfacial strength and the area under the stress-displacement curve which is considered as the critical energy release rate G_c [12]. Different forms of traction-separation law are given in Fig. 1 in which δ^0 denotes the critical opening displacement for damage onset. A linear elastic-linear softening behaviour is generally selected to be implemented in the simulations [5].

To illustrate the cohesive zone formation and transition, a DCB test can be considered as shown in Fig. 2. After obtaining its maximum interfacial strength the element next to crack tip enters to its softening region of the traction-separation response. Ahead of the crack tip a cohesive zone length, l_{CZ} , forms as the opening stress increases and more elements undergo the same procedure. $l_{CZ,f}$ is defined as the maximum l_{CZ} when the first cohesive element at crack tip fails completely and causes the crack propagation [13].

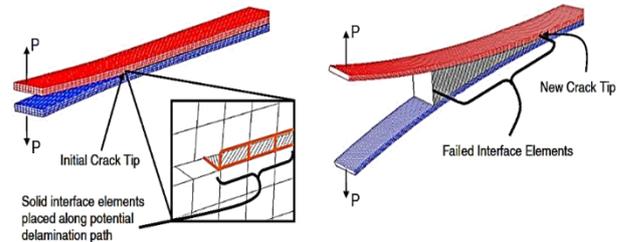
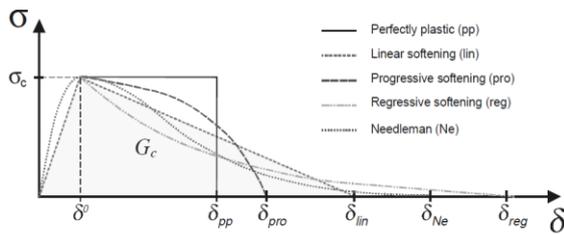


Figure 1. Various Forms of Traction-Separation Curves [5] **Figure 2.** Cohesive Element Transition and Delamination Path in DCB [13]

One of the drawbacks of the CZM is the mesh size requirements. In addition, the interface parameters are required to be selected precisely. Various studies have been conducted regarding the effect of different parameters on behaviour of the cohesive elements. Turon et al. [8] investigated the minimum penalty stiffness (K) and minimum number of elements within the Cohesive Zone to correctly capture the delamination phenomena in FEM. It is observed that stiffness parameter is required to be large enough to ensure a stiff connection between two adjacent plies and small enough to avoid oscillatory and incorrect results. The element size is considered to be smaller than l_{CZ} (distance between the crack tip and the element with maximum traction). Cohesive zone length can be predicted by using:

$$l_{CZ} = ME \frac{G_c}{(\tau^0)^2} \quad (1)$$

where E is the Young modulus (in the case of transversely isotropic materials, transverse elastic modulus, E_2). G_c is the critical energy release rate, τ^0 is the maximum interfacial strength and the value for M parameter depends on the selected cohesive zone model. According to the study of Hillerborg et al. [11], M is selected as 1. The minimum number of elements, N_e , required in cohesive zone to obtain more accurate stress distribution within this zone in FEM, is not well addressed. Falk et al. [14] have

assigned 2-5 elements and Turon et al. [8] have suggested 3 elements. This leads us to calculate the length of the cohesive zone element, l_e , using:

$$N_e = \frac{l_{CZ}}{l_e} \geq 3 \quad (2)$$

To eliminate this computationally inefficient barrier caused by assigning super fine mesh, Turon suggests the reduction of maximum interfacial strength to enlarge the cohesive zone length. The value of the interfacial strength, $\bar{\tau}^0$, required for desired number of elements within the cohesive zone, N_e^0 , is derived from Eq. 1 and Eq. 2:

$$\bar{\tau}^0 = \sqrt{\frac{EG_c}{N_e^0 l_e}} \quad (3)$$

The interfacial strength then selected as:

$$T = \min\{\tau^0, \bar{\tau}^0\} \quad (4)$$

Fracture toughness and strength values (G_c and τ^0) are based on the loading mode. Satisfying results have been captured with reduced interfacial strength and a coarser mesh when compared to using default nominal strength. The main shortcoming of this approach is the inaccuracy of predicted stress distribution in the vicinity of the crack tip.

In this study first, following Turon's approach [8], the effect of mesh size, interface stiffness, interface strength, and the selection of the parameters were investigated for models with 2D plane strain, 3D continuum shell, and 3D solid elements by analyzing the Mode I delamination test (DCB) for unidirectional T300/977-2 carbon-fiber-reinforced epoxy laminate. Later, Cohesive Zone Modelling parameters for unidirectional AS4/8552 carbon-fiber-reinforced epoxy laminate is optimized in order to predict the delamination behaviour in Mode I. These parameters can then be used to model the progressive behaviour of more complex specimens and structures made of AS4/8552.

3. Numerical benchmarking study for T300/977-2 carbon-fiber-reinforced epoxy laminate

In this part, a benchmarking study is carried out using the same material and modelling approach that Turon [8] adopted in his analysis of DCB test using 2D plane strain elements (CPEG4). However, the approach is carried forward by implementing it to 3D models using solid (C3D8) and continuum shell elements (SC8R), which should be used to model progressive failure of composite structures.

3.1. Benchmarking model with plane strain elements with cohesive surface method

DCB test is simulated by using generalized plane strain elements and applying interaction properties between two cantilevers. K is calculated as 270000 N/mm^3 in the study of Turon et al. [8] assuming that at least 4 elements should be placed in cohesive zone. Cohesive zone length is calculated approximately as 0.95 mm which means that element length must be at least 0.25 to reach accurate results. Fig. 5 shows the FE model mesh structure (=1mm) and applied boundary conditions. Fig. 6 illustrates the deformed shape of the 2D DCB. Fig. 7 shows the load-displacement results of the proposed model with several mesh sizes. These plots are also compared to the experimental data given in [8]. All models have a constant interfacial strength of 60 MPa.

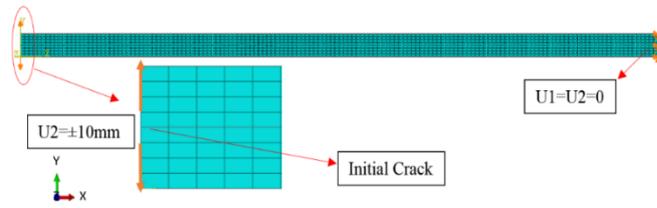


Figure 5. Mesh and boundary conditions for 2D DCB model

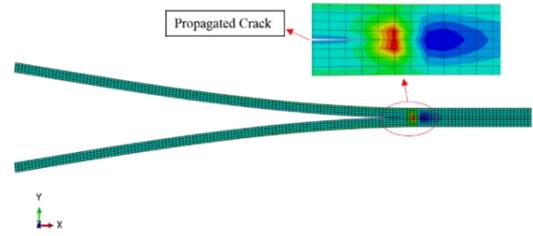


Figure 6. Deformed state of the 2D DCB model

According to the results, there is no considerable difference for element lengths $L_e = 0.05, 0.25$ and 0.5 mm, which means using 2-5 elements for the cohesive zone would give satisfying results (with constant interfacial strength) and decreasing mesh size would have negligible effect on results. In this case having 4 elements in the cohesive zone would be adequate in our simulations. Despite the decrease in mesh size there still exist some differences with the experimental result. Since the interfacial stiffness parameter (K) is carried out by using Eq. 7:

$$K = \frac{\alpha E_3}{t} \quad (7)$$

where α is a parameter much greater than 1 ($\alpha \gg 1$) and t is the thickness of the sub-laminate, α can be modified to have a better correlation. Considering another case where larger mesh size is preferred, the requested cohesive zone length needs to be 4 mm. So the interfacial strength needs to be artificially reduced by using Eq. 3. Mesh size of 1 mm can be applied with $\bar{\tau}^0 = 32$ MPa. The load displacement curves obtained using $\bar{\tau}^0 = 32$ MPa and $\tau^0 = 60$ MPa are compared with experimental curve in Fig. 8. Since the number of cohesive elements has increased by using $\bar{\tau}^0$, predictions are in a good correlation with experimental data. The value for α is 50 in these simulations. The peak load can be reduced by using a smaller value for α and similarly it can be increased by a higher α .

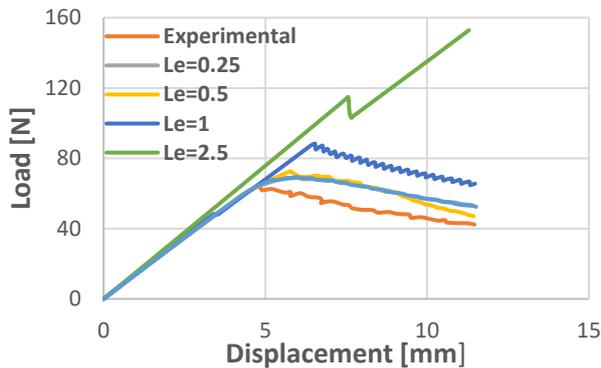


Figure 7. Load-displacement curve for 2D DCB with various mesh sizes

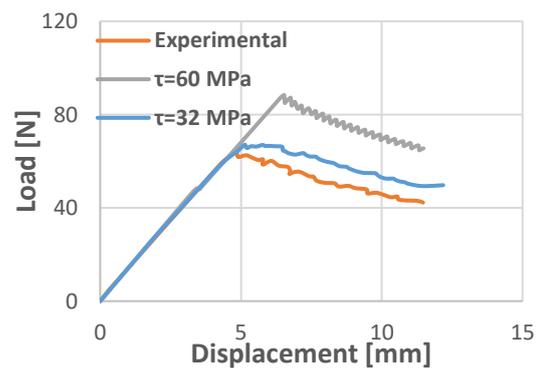


Figure 8. Load-displacement curve for DCB using reduced interface strength

3.1.1 Solid/continuum shell elements with cohesive surface interaction

The model for 3D DCB has been investigated by using solid elements and also continuum shell elements which are 8-node quadrilateral in-plane elements. The load-displacement curves obtained with these two 3D approaches have been compared with experimental and 2D model results in Fig. 9. Stiffness parameter (K) in a continuum shell based model has been reduced to 100000 N/mm^3 in order to observe the peak load reduction. It can be observed that the 2D model approximates to the experimental curve quite closely and the model using continuum shell elements provide acceptable predictions.

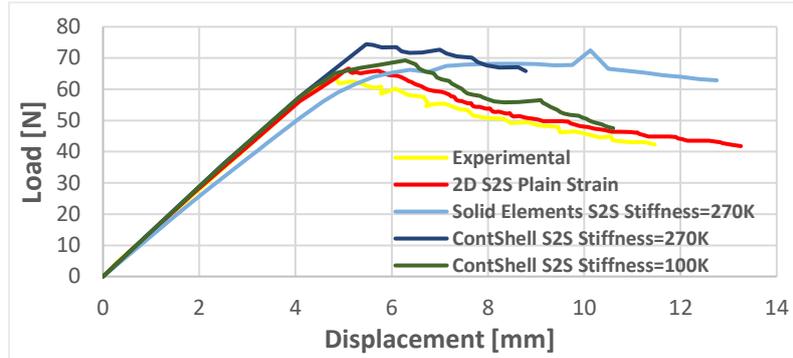


Figure 9. Load-displacement curves for various DCB models

3.1.2. 3D DCB using continuum shell elements with cohesive elements

Final model, shown in Fig. 10 is based on cohesive elements rather than cohesive surfaces. The difference with the previous model is that only one deformable 3D extruded part is created including the initial crack and two cantilever arms and a 0.001 mm thick cohesive section in between to assign cohesive material properties. The other parameters such as meshing technique and boundary condition are the same as previous models. Stiffness degradation parameter SDEG is plotted in Fig. 11 for crack initiation and final state after crack propagation. The red color refers to delamination, where the value of the SDEG parameter reaches 1.

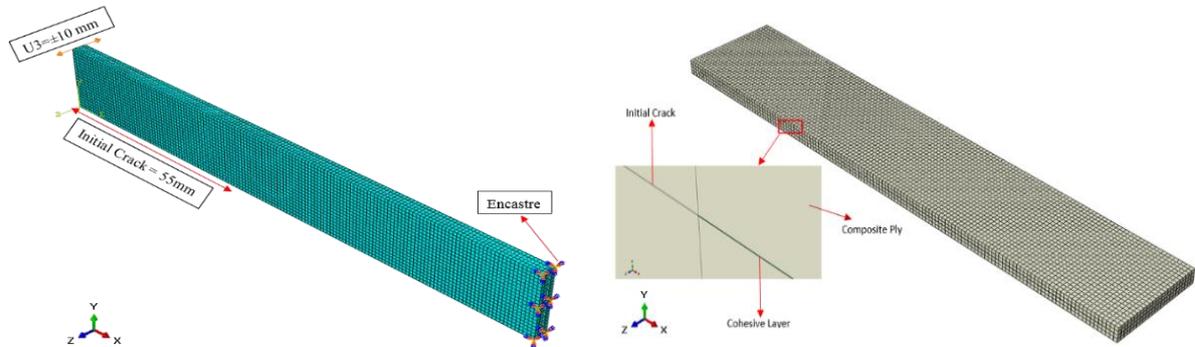


Figure 10. 3D Finite Element model of DCB specimen using cohesive elements

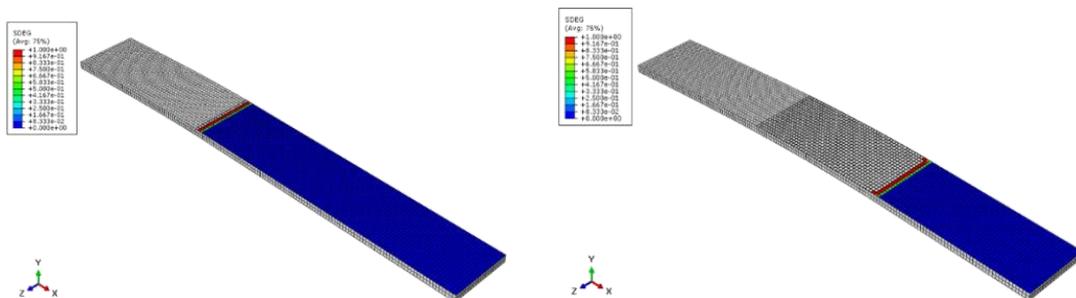


Figure 11. Delamination initiation (left) and extension (right) in 3D DCB (lower arm) using continuum shell elements and cohesive elements

The load-displacement curves based on this approach and those of solid elements and continuum shell elements in previous models are compared with 2D approach and experiments in Fig. 12. Using CZE and continuum shell elements to simulate the DCB test gives better results compared to using solid elements.

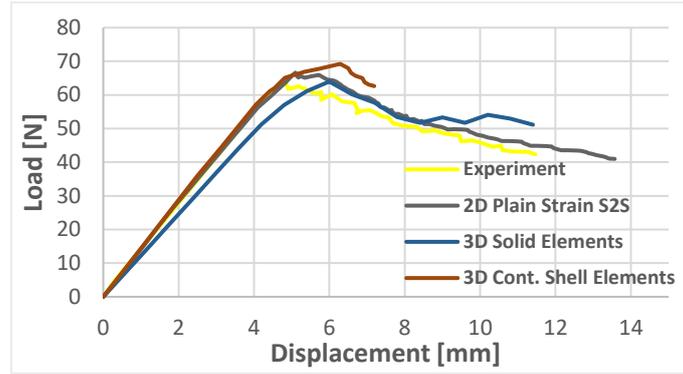


Figure 12. Load-displacement curves for DCB models using cohesive elements for delamination

4. Experimental study for AS4/8552 carbon-fiber-reinforced epoxy laminate

A 300 x 300 mm² AS4/8552 UD laminate consist of 20 plies and a teflon insert between 10th and 11th plies are manufactured according to the Manufacturer's Recommended Cure Cycle (MRCC) in the autoclave. Then the plate is cut with a water-cooled diamond saw into desired specimen dimensions. The DCB tests are performed according to ASTM D5528-13 test standard [15]. Then, test results are compared with 2D and 3D Finite Element model. Test setup and specimen configuration is shown in Fig. 3 and Fig. 4.

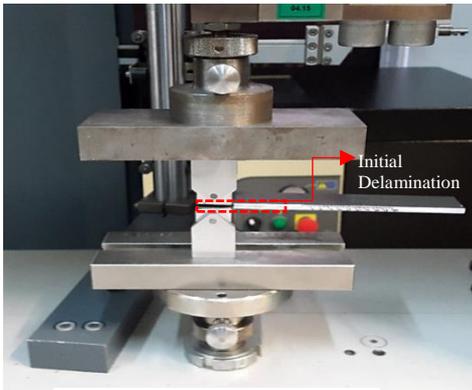


Figure 3. Test setup

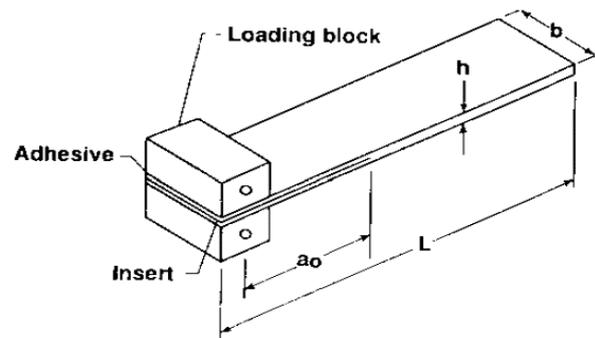


Figure 4. DCB specimen with loading blocks [15]

Where a_0 is the initial delamination length, b is the width, h is the thickness and L is the length of the DCB specimen. Modified Beam Theory (MBT) and a Modified Compliance Calibration method (MCC) can be used to calculate interlaminar fracture toughness as given in Eq. 5 and Eq. 6, respectively:

$$G_I = \frac{3P\delta}{2ba} \quad (5)$$

$$G_I = \frac{3P^2 C^{2/3}}{2A_1 b h} \quad (6)$$

Where P is the applied load, δ is load point displacement, a is defined as delamination length, C is the compliance (δ/P), of DCB specimen and A_1 is the slope in plot of a/b versus $C^{1/3}$. The Mode I fracture toughness results by using these two equations and load-displacement plots for three specimens tested are given in Table 1.

Table 1. Experimental results with calculated Mode I fracture toughnesses

	Width (mm)	Thickness (mm)	Max. Load (N)	Disp. at Max Load (mm)	K (N/mm)	C (mm/N)	G_{Ic} (MBT) (KJ/m ²)	G_{Ic} (MCC) (KJ/m ²)
Sp#1	24.25	3.50	60.56	3.46	14.98	6.6773E-02	0.26	0.30
Sp#2	24.25	3.68	53.27	4.32	10.69	9.3449E-02	0.28	0.28
Sp#3	24.50	3.54	51.64	3.80	12.23	8.1705E-02	0.24	0.25
Average							0.26	0.28

The same 2D and 3D FE models used in benchmarking study are customized for AS4/8552 with using $G_{Ic}=0.28$ KJ/m² and the material properties given in Table 3 and Table 4. Element size of 1 mm with reduced interfacial strength by using Eq. 2 (= 26.46 MPa) has been selected. Fig. 13 represents the load-displacement responses of the experimental tests and 2D and 3D FE models.

Table 3. Material Parameters for UD Composite [16]

E_1 (GPa)	E_2 (GPa)	E_3 (GPa)	G_{12} (GPa)	G_{23} (GPa)	ν_{12}	ν_{23}
142.84	10	9.5	5.571	3.278	0.263	0.525

Table 4. Interlaminar strength, fracture energy and stiffness parameters for cohesive elements

σ_{33c} (Mpa)	σ_{13c} (Mpa)	G_{Ic} (KJ/m ²)	G_{IIc} (KJ/m ²)	K_{nn} (N/mm ³)	K_{ss} (N/mm ³)
81	114	0.28	2.59	2.72E+006	2.72E+006

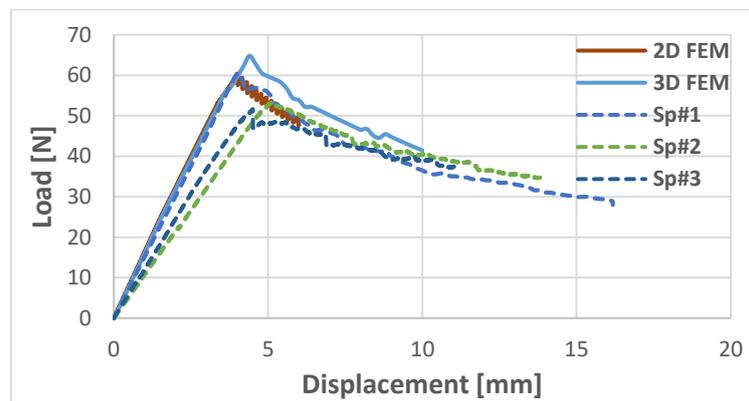


Figure 13. Load-displacement responses of experimental tests, 2D and 3D FEM models

5. Conclusions

In order to assign coarser meshes in modelling Mode I delamination behaviour of AS4/8552 carbon-epoxy laminates, an approach based on an artificial increase of the cohesive zone length by lowering the interfacial strength is followed. In this paper, DCB tests are performed experimentally to obtain the load-

displacement curve and to evaluate the mode I interlaminar fracture toughness value. Moreover, 2D and 3D numerical models are applied to simulate DCB test within ABAQUS. A parametric study is carried out for both models to investigate the effect of various cohesive parameters. Accuracy of the results shows that propagation of delamination can be accurately predicted with coarser meshes using this computational-time-friendly approach.

References

- [1] G. Alfano and M. A. Crisfield. Finite element interface models for the delamination analysis of laminated composites: mechanical and computational issues. *International Journal for Numerical Methods in Engineering*, 50:1701–1736, 2001.
- [2] Y. Mi, M.A. Crisfield and G.A.O. Davies. Progressive delamination using interface elements. *Journal of Composite Materials*, 32:1246–1272, 1998.
- [3] A. Needleman. A continuum model for void nucleation by inclusion debonding. *Journal of Applied Mechanics*, 54:525–532, 1987.
- [4] V.K. Goyal, E.R. Johnson, C.G. Dávila and N. Jaunky. Irreversible constitutive law for modeling the delamination process using interfacial surface discontinuities. *Composite Structures*, 64:91–105, 2004.
- [5] P.P. Camanho, C.G. Dávila and M.F. de Moura. Numerical simulation of mixed-mode progressive delamination in composite materials. *Journal of Composite Materials*, 37:1415–1424, 2003.
- [6] A. Turon, P.P. Camanho, J. Costa and C.G. Dávila. A damage model for the simulation of delamination in advanced composites under variable-mode loading. *Mechanics of Materials*, 38:1079–1089, 2006.
- [7] R. Krueger. The virtual crack closure technique: history, approach and applications. *NASA/Contractor Report 2002-211628*, 2002.
- [8] A. Turon, C.G. Davila, P.P. Camanho and J. Costa. An engineering solution for mesh size effects in the simulation of delamination using cohesive zone models. *Engineering Fracture Mechanics*, 74:1665–1682, 2007.
- [9] G. I. Barenblatt. The mathematical theory of equilibrium cracks in brittle fracture. *Advances in Applied Mechanics*, 7:55–129, 1962.
- [10] D.S. Dugdale. Yielding of steel sheets containing slits. *Journal of the Mechanics and Physics of Solids*, 8:100–104, 1960.
- [11] A. Hillerborg, M. Modeer and P.E. Petersson. Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite elements. *Cement and Concrete Research*, 6:773–782, 1976.
- [12] K. Song, C. Davila and C. Rose. Guidelines and parameter selection for the simulation of progressive delamination. *ABAQUS User's Conference*, 1–15, 2008.
- [13] P.W. Harper and S.R. Hallett. Cohesive zone length in numerical simulations of composite delamination. *Engineering Fracture Mechanics*, 75:4774–4792, 2008.
- [14] M.L. Falk, A. Needleman and J.R. Rice. A critical evaluation of dynamic fracture simulations using cohesive surfaces. *Journal de Physique*, 4:43–50, 2001.
- [15] ASTM D5528-13. Standard test method for Mode I interlaminar fracture toughness of unidirectional fiber-reinforced polymer matrix composites. *American Society for Testing and Materials*, 1:1–12, 2013.
- [16] F.E. Öz. Micromechanical progressive damage model for predicting resin dominated strength values of fibre reinforced composites under various types of loading. *M.Sc. Thesis*, Boğaziçi University, 2012.