Original scientific paper UDK 539:620.1>:624.07
https://doi.org/10.32762/zr.23.1.2

MODELLING ADHESIVE USING NON-LINEAR TRUSS ELEMENTS IN MODE I DELAMINATION PROBLEMS

MODELIRANJE LJEPILA KORISTEĆI NELINEARNE ŠTAPNE ELEMENTE KOD PROBLEMA RASLOJAVANJA U MODU I

Maedeh Ranjbar*, Gordan Jelenić*, Leo Škec*

Abstract

A fast and simple finite element model is presented in this paper to simulate the crack propagation in notched beam structures with two layers and one interface medium in Mode I delamination. The truss elements from FEAP element library are endowed with a user material law describing the bilinear cohesive zone model (CZM) with the material unloading path defined by embedding a history variable in the response. The layer is modelled with linear elastic Timoshenko beam elements. The method is used for damage growth and damage propagation simulations in interfaces with both ductile and brittle materials. The method is robust for ductile interfaces, but for brittle interfaces more specific numerical techniques are required. The results are evaluated using available analytical and numerical solutions and a good agreement is achieved.

Key words: *DCB test, Crack propagation, Cohesive Zone Model, FEAP implementation*

Sažetak

U ovome radu predstavljen je brz i jednostavan model konačnih elemenata za opis raslojavanja u modu I u zasječenim dvoslojnim grednim konstrukcijama s kontaktnim materijalom među slojevima. Element rešetke iz knjižnice elemenata programa FEAP obogaćen je korisničkim materijalnim modelom koji opisuje bilinearni model kohezivne zone, a u kojem je rasterećenje omogućeno definiranjem varijabli koje pamte povijest odgovora. Gredni sloj opisan je linearnim Timoshenkovim grednim

* University of Rijeka, Faculty of Civil Engineering, Radmile Matejčić 3, 51000 Rijeka
E-mail: {maedeh.ranjbar, Gordan.jelenić, Leo. škec}@gradri.uniri.hr

Rad zaprimljen: 28.07.2020. Rad prihvaćen: 28.10.2020.

elementom. Metodu koristimo za opis razvoja oštećenja u kontaktnom sloju, koji može biti izrađen od duktilnog ili krtog materijala. Metoda je robusna za duktilne kontaktne materijale, a krti materijali zahtijevaju drugačiji pristup, prilagođen takvim materijalima. Rezultati su ocijenjeni usporedbom s raspoloživim analitičkim i numeričkim rješenjima s kojima je uočena dobra podudarnost.

Ključne riječi *test dvostruke konzole, razvoj pukotine, model kohezivne zone, ugradnja u FEAP*

1. Introduction

Layered structures provide an extremely effective means of optimizing functional and structural performance of diverse mechanical systems in many engineering applications. Composite structures are made of two or more components from different materials combined in such a way that each of them fulfils the function for which its material characteristics are best suited. The mechanical behaviour of these structures largely depends on the type of connection between the layers. Delamination is one of the most prevalent and severe failure modes in layered composite structures, and has been widely investigated, from both experimental and numerical standpoints. The crack can propagate in the delamination problem in the opening (Mode I), sliding (Mode II) or tearing (Mode III) modes. In this paper we focus only on Mode I delamination and the double cantilever beam (DCB) test, which is the standard test for determining fracture toughness G_{IC} in Mode I [1,2].

Fracture mechanics and damage mechanics are two main approaches for the numerical simulation of delamination. In the frame-work of damage mechanics, the most prevalent models for describing the interlayer behaviour are cohesive-zone models (CZMs), originally proposed by Dugdale [3] and Barenblatt [4]. As damage is usually not introduced in the layers, they are described through a simple Hook's material law. Plane-strain elements in assembly with interface elements have been traditionally used to simulate crack propagation in delamination tests, including single and mixed modes [5, 6]. Modelling delamination using beam FEs rather than 2D plane–strain FEs, has been recently proposed to improve the computational efficiency of the numerical simulation [7]. For DCB tests, analytical solutions are also available, in fact, not only for linear-elastic analysis, but also for a bi-linear constitutive law [8] as one of the most common traction-separation laws [9].

Oscillations in the load-displacement curve and the related convergence problems are usually reported in the numerical analysis of delamination tests, including DCB tests [5,7,10]. In DCB tests, the oscillations may be reduced by simply refining the FE mesh [5, 7] and using Simpson integration rule instead of Gauss-integration [11]. In more complex delamination tests, snap-through and snap-backs in the response of a delaminated composite can be successfully handled using the arc-length method [12, 13]. Moreover, the interface parameters play a fundamental role in the resulting oscillations and instabilities: the more ductile an interface, the more robust the solution. The solution is strongly dependent on the tensile strength, but a good agreement between the CZM model and the linear-elastic fracture-mechanics (LEFM) formulation in low values of tensile strength can still be obtained.

The present paper proposes a very simple model for the numerical simulation of delamination in Mode I for the DCB tests. A symmetric model, where only one half of the DCB specimen is modelled, employs beam (Frame) and truss (Bar) elements from the FEAP [14] element library for the arm and the interface, respectively. User material input and user material law subroutines are required to enhance the original truss elements from the FEAP library with a bilinear cohesive-zone model. Thus, once that the material model is modified, such truss elements can be easily implemented in a simulation using the standard FEAP framework. However, such model is not applicable for simulations of Mode II delamination problems.

The outline of the paper is as follows. In Section 2, the problem and its governing equations are described. In Section 3, the numerical method is explained from the points of view of implementation and finite element modelling. The numerical results are reported in Section 4 for ductile, and brittle interfaces and compared to available analytical and numerical results.

2. Problem Description

The geometry and boundary restraints of the problem are depicted in Figure 1 for a moment-loaded DCB (MDCB) with prescribed rotations and for a standard force-loaded DCB with a prescribed displacement, respectively. The specimen has a length L, width b and thickness 2h. In contrast to DCB, MDCB belongs to the family of steady-state J-integral specimens and is well-known in the field of experimental and analytical fracture mechanics [15, 16]. In this paper, results for both DCB and MDCB are presented.

The arms are modelled using simple Hook's material law ($\sigma = E\varepsilon$), while the interface is modelled using a bi-linear constitutive law shown in Figure 2. The interface material has three parameters, namely σ_{max}, δ_0 and δ_C as the maximum value of contact tractions, the relative displacement at the linear-elastic limit and the relative displacement for which a total loss of adhesion in the interface takes place, respectively. Parameters σ_{max} and δ_0 define the elastic limit (point C in Figure 2), while at a point B, closer to the crack tip, there is an ongoing damage growth. At the crack tip (point A), the adhesion is completely lost and the crack starts to propagate. The cohesive-zone model can be formulated as:

$$\sigma = \begin{cases} S\delta & if\ \bar{\beta} \leq 0 \\ (1-g)S\delta & if\ \bar{\beta} > 0 \end{cases}. \tag{1}$$

in which the damage parameter is defined as

$$\bar{\beta}(\tau) = \max_{0 \leq \tau' < \tau} \beta(\tau'), \tag{2}$$

$$\beta(\tau') = \frac{\delta}{\delta_0} - 1, \tag{3}$$

the stiffness of the interface is

$$S = \frac{\sigma_{max}}{\delta_0}, \tag{4}$$

and the damage parameter reads

$$g = \min\left\{1, \frac{\delta_c}{\delta_c - \delta_0}\frac{\bar{\beta}}{1+\bar{\beta}}\right\}, \tag{5}$$

where σ and δ denote the traction and separation of the interface, respectively.

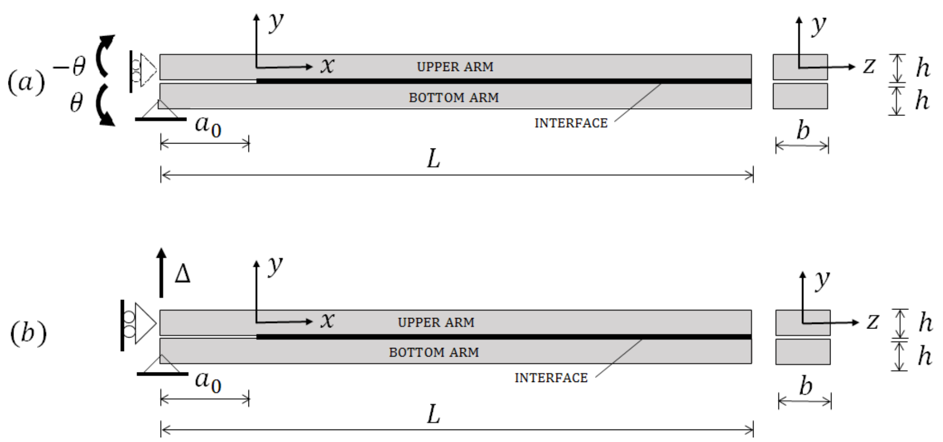

Figure 1. *Geometry and boundary conditions of a DCB specimen with, a) prescribed rotations (MDCB) and b) prescribed displacement*

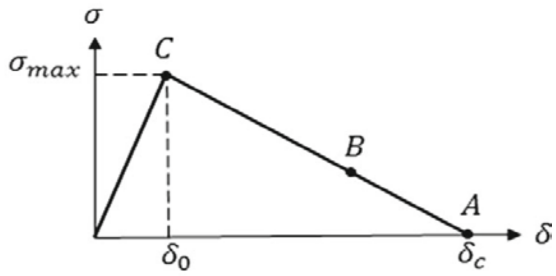

Figure 2. *A typical bi-linear CZM representing interface constitutive behaviour*

3. Numerical Simulation of the problem

3.1. FEAP implementation of the CZM law

As shown in equation (1), the constitutive equation of the interface relates the stress to the relative displacement, which is unlike the common material laws, where strains are related to stress values in the constitutive equations. In FEAP, the residual and tangent stiffness computations are based on the constitutive stress-strain relations and the existing truss element can be adapted to account for the non-linear constitutive law (1). This is performed via a user material input subroutine UMATI with three material inputs (σ_{max}, δ_0 and δ_C) and one history variable ($\bar{\beta}$) implemented in FEAP. The implemented user material law is called in the solution procedure and computes the components of stress based on equation (1) and tangent stiffness value (dd) using the following relations:

$$dd = \begin{cases} S & if\ \bar{\beta} \leq 0 \\ (1-g)S & if\ g < 1\ and\ \beta < \bar{\beta} \\ \frac{\sigma_{max}}{\delta_0 - \delta_c} & if\ g < 1\ and\ \beta > \bar{\beta} \\ 0 & if\ g = 1 \end{cases} . \tag{6}$$

The material behaviour of a single non-linear truss element is shown in Figure 3, where loading, unloading and reloading parts can be observed. The cross-section of the truss element is set to 1mm^2 its length is 1 mm and σ_{max}, δ_0 and δ_C are equal to 10 MPa, 0.5 mm and 2 mm, respectively.

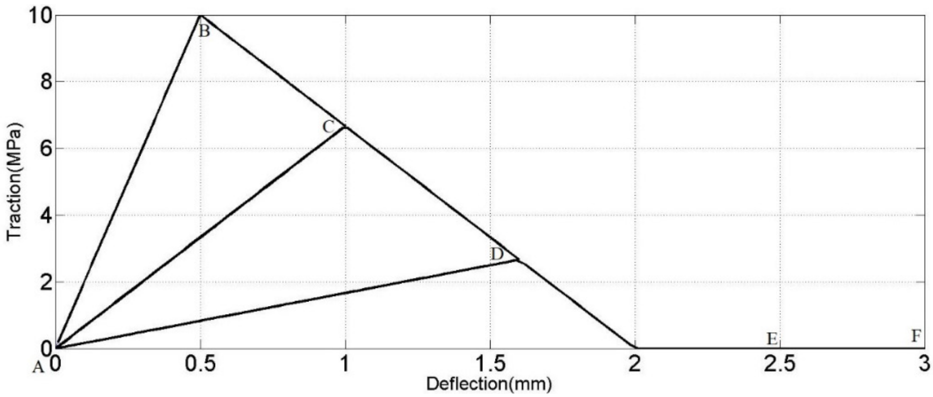

Figure 3. *Unloading-reloading paths (CA-AC and DA-AD) of the implemented Truss Element with a bi-linear CZM law*

3.2. Finite Element Model of the problem

Due to the symmetry of the structure, only a half of it is modelled using finite elements as shown in Figure 4. The beam is divided in n Timoshenko beam (Frame) elements of equal length. The interface is modelled by connecting $(n(1 - a_0/L)+1)$ truss elements to each beam node in the interface area if n is set in such a way that $\frac{a_0 n}{L}$ is an integer. Essential boundary conditions are applied on node $(n + 2)$ to node $(n(2 - a_0/L)+2)$ by fixing them to satisfy the symmetry boundary condition. Depending on the DCB test simulated, vertical displacement or angular rotation are prescribed at node $(n + 1)$ at the right-hand end of the beam. The cross-sectional area of each truss is computed in a way that the total surface of the interface is uniformly distributed between the truss elements, i.e.

$$A_{truss} = \frac{b \times (L - a_0)}{n} \tag{7}$$

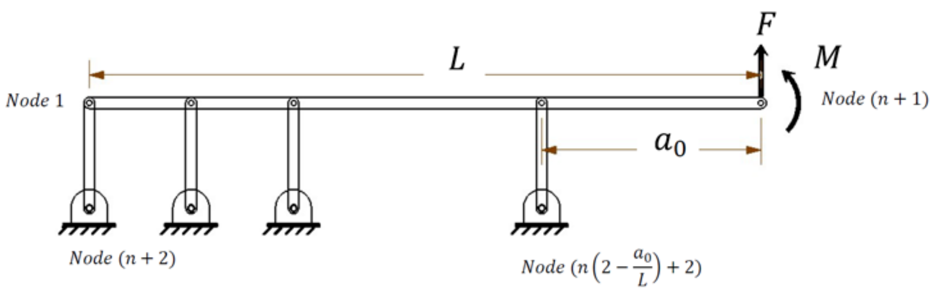

Figure 4. *Finite element modelling of the problem*

4. 4. Numerical Results

In this section, three numerical examples, for which analytical or numerical results are already available in the literature, are used to test the present numerical model.

4.1. Example 1

A DCB test with variable interface properties, but equal area under the traction-separation law , is simulated. The analytical solution is obtained by assuming an infinite length of the beam [8], but setting L = 200mm is sufficient for excluding the end-of-beam effects in the biggest part of the numerical simulation. Referring to Figure 1, a DCB with dimensions h = 6 mm, b = 25 mm and a_0 = 30 mm is considered. Table 1 lists the material properties of the bulk material (beam layers) and interface. The maximum contact traction is varied between 7.5 and 120MPa to provide 5 cases of different brittleness in the interface.

Table 1. *Material properties of the DCB example 1*

BULK		INTERFACE		
E (GPa)	ϑ	σ_{max} (MPa)	δ_c (mm)	δ_0 (mm)
70	0.33	7.5	0.26	$0.01\,\delta_c$
		15	0.13	
		30	0.066	
		60	0.033	
		120	0.016	

2000 2-noded Timoshenko linear beam (Frame) elements from FEAP element library are distributed over the upper half of the DCB, which means that the distance between the nodes is l_e = 0.1 mm. Therefore, according to Figure 4, there are1701 truss elements at the interface, each of them connected to a Frame-element node. Such mesh proved to be sufficiently fine to avoid oscillations and convergence issues typical for FE simulations delamination problems [7]. Due to symmetry, the values of δ_0 and δ_c are divided by 2 in the FEAP input file. A vertical deflection $\Delta/2$ = 10 mm is applied on the left-end node in 250 increments. Do note that, because of the symmetry of the DCB test, when only one half of the specimen is modelled, the deflection of the arm is half of the total value of the crack mouth opening Δ (see Figures 1(b) and 4). The FEAP simulation completes with converged Newton-Raphson iterations with a quadratic rate of convergence and a maximum of 6 iterations per load step.

The crack length can be measured during crack propagation by detecting the location of leftmost truss element with zero stiffness. The agreement between FEAP and analytical results, which are obtained from the software EasyDCB [8], is shown in Figure 5. Small differences between the solutions can be noticed only for relatively large values of crack length because of the infinite-length assumption in the analytical solution. In Figure 6, the force-displacement curve for the DCB test is obtained from both FEAP simulation and EasyDCB. An excellent agreement is observed between proposed interface model and the analytical solution.

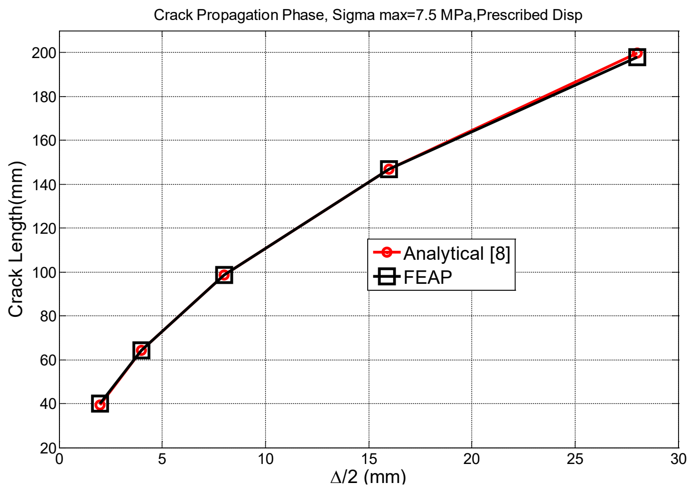

Figure 5. *Crack length development with respect to the prescribed displacement in the DCB example 1 for the case with*

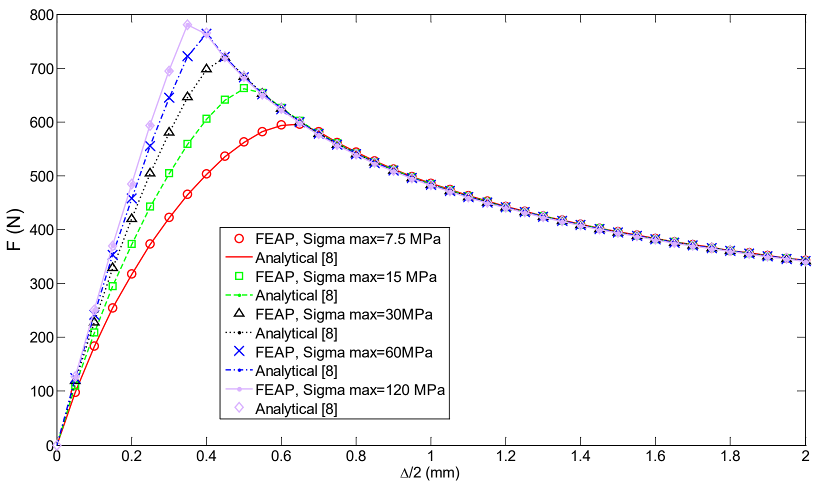

Figure 6. *Load-displacement plots of the DCB example 1 with variable interface properties*

4.2. Example 2

Using the same material properties, geometrical dimensions and finite element descriptions as in the previous example, in this example a MDCB test is simulated. Rotation is applied on the left end node and the load-displacement results are plotted in Figure 7, which shows an excellent agreement with the analytical results [8].

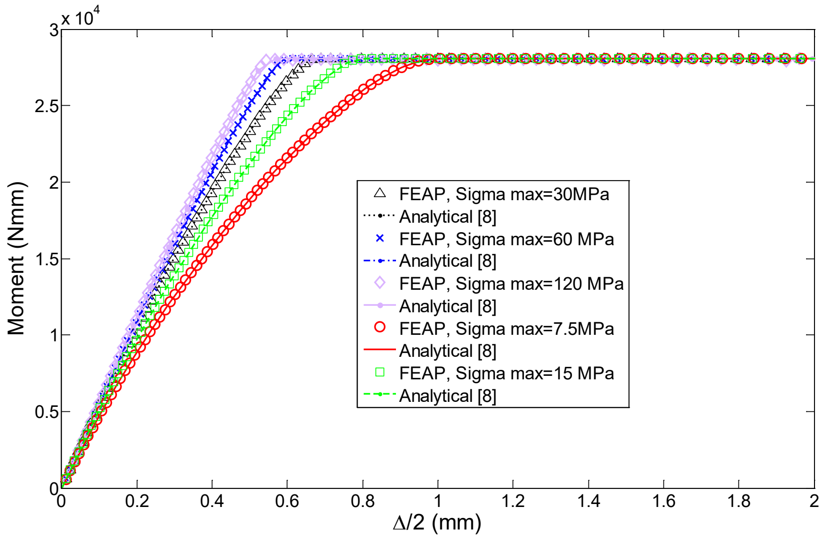

Figure 7. *The load-displacement plots of the MDCB example 2*

4.3. Example 3

The specimen used in this example is shown in Figure 8 with its corresponding geometrical properties (the width of the beam is 20 mm), boundary conditions and loading. The material properties for bulk and interface are listed in Table 2. 800 2-noded Timoshenko beam (Frame) elements are used to discretise the DCB arm, which means that the distance between the nodes is . Thus, according to Figures 4 and 8, at the interconnection there are 561 nodes to which truss elements are attached. The solution procedure is not as robust as the previous examples due to the high brittleness of interface material. Very small increments and a relatively large number of iterations in the displacement control solution are required to avoid loss of convergence in the Newton-Raphson solution procedure. In particular, the displacement increment is decreased from 0.04 mm in the previous examples to 0.005 mm, while the maximum number of iterations is increased from 7 to 15. To simulate brittle interface numerically a more sophisticated solution algorithm than the displacement control method is needed (not included in the present paper). The numerical results are presented in Figure 9 and are compared to the multi-layer beam model from [7]. The results are in good agreement, oscillations are less pronounced in the truss element model results but more differences after the limit point exist due to reducing the residual tolerance to avoid losing convergence near the limit point (the point with singularity in the tangent stiffness of the finite element method).

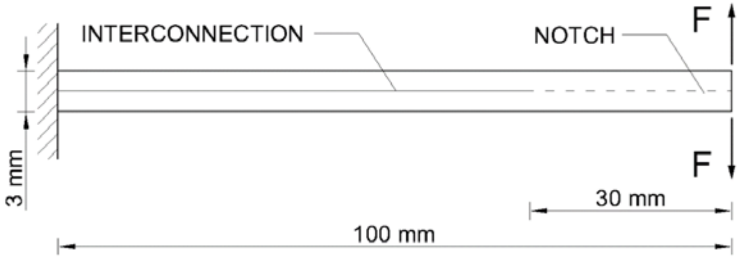

Figure 8. *Geometrical properties of specimen example 3*

Table 2. *The material properties of the DCB in example 3*

BULK		INTERFACE		
E (GPa)	*G(GPa)*	σ_{max} (MPa)	δ_c (mm)	δ_0 (mm)
135.3	5.2	57	0.009825	1e-7

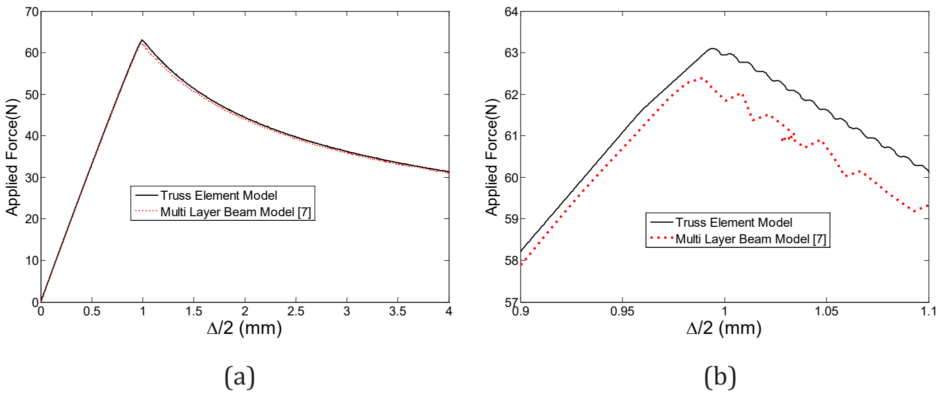

Figure 9. *The load-displacement plot of DCB in example 3, a) A range of interest, b) zoom*

5. Conclusion and Outlook

In this paper, a simple and efficient model for simulation of crack propagation in Mode I using the DCB test is presented. A truss element with a non-linear material law based on the bi-linear CZM can model the interlaminar media in the composites in a simple manner. The model is accurate, fast and robust for the interfaces with ductile interfaces but it is recommended for the brittle interfaces only if decreasing σ_{max} is feasible. Numerical instabilities, typical of numerical simulations with cohesive-zone elements, can be observed in the numerical simulation of brittle interfaces with non-linear truss elements. However, this can be suppressed by refining either the mesh or increment sizes. The material behaviour for interface media is implemented in FEAP, and the simulation results are validated against available analytical and numerical results.

The lack of efficiency of the non-linear truss elements in the case of brittle interfaces is the main motivation to implement a 2D interface element in FEAP in the next step of this research to simulate continuous interface. Besides that, with the coupled interface element [5], Mode II and mixed-mode simulations will be possible. Efficiency and robustness of this extension will be assessed and reported subsequently.

Acknowledgements. The results of this paper have been obtained within the research project IP-2016-06-4775: *"Assumed Strain Method in Finite Elements for Layered Plates and Shells within the Application on Layer Delamination Problem-ASDEL"* financially supported by the Croatian Science Foundation from 2017.

References

[1] Anderson T L. (2005) ''Fracture Mechanics: Fundamentals and Applications'', Boca Raton. FL: Taylor & Francis.

[2] ASTM D5528-01 (2007) ''Standard Test Method for Mode I Interlaminar Fracture Toughness of Unidirectional Fiber-Reinforced Polymer Matrix Composites'', West Conshohocken (PA): ASTM International.

[3] Dugdale DS. (1960) ''Yielding of Steel Sheets Containing Slits'', J Mech Phys Solids. 8, 100–104.

[4] Barenblatt GI. (1959) ''The Formation of Equilibrium Cracks during Brittle Fracture - General Ideas and Hypothesis, Axially Symmetric Cracks'', J Appl Math Mech. 23(3), 622–636.

[5] Alfano G, Crisfield MA. (2001) ''Finite Element Interface Models for the Delamination Analysis of Laminated Composites: Mechanical and Computational Issues'', Int J Numer Methods Eng. 50(7), 1701–1736.

[6] Mi Y, Crisfield MA., Davies GAO, Hellweg HB. (1998) ''Progressive Delamination Using Interface Elements'', J Compo Mater. 32(14),1246-1272.

[7] Škec L, Jelenić G, Lustig N. (2015) ''Mixed-Mode Delamination in 2D Layered Beam Finite Elements'', Int J Numer Methods Eng. 104, 767–88.

[8] Škec L., Alfano G., Jelenić G. (2019) ''Complete Analytical Solutions for Double Cantilever Beam Specimens with Bi-linear Quasi-brittle and Brittle Interfaces'', Int. J. Fract. 215. 1–37.

[9] M.J. van den Bosch, P.J.G. Schreurs, M.G.D. Geers. (2006) ''An Improved Description of the Exponential Xu and Needleman Cohesive Zone Law for Mixed-Mode Decohesion'', Eng Fract Mech. 73(9), 1220-1234.

[10] Blackman BRK, Hadavinia H, Kinloch AJ, Williams JG. (2003b) ''The Use of a Cohesive Zone Model to Study the Fracture of Fibre Composites and Adhesively-bonded Joints'' Int J Fract. 119, 25–46.

[11] Schellekens JCJ, De Borst R. (1993) ''On the Numerical Integration of Interface Elements'', Int J Solids Strut. 36, 43-66.

[12] Crisfield MA. (1981) ''A Fast Incremental/Iterative Solution Procedure that Handles Snap through'', Comput Struct. 13, 55-62.

[13] Hellweg HB, Crisfield MA. (1998) ''A New Arc-Length Method for Handling Sharp Snap-Backs'', Comput Struct. 66(5), 704-709.

[14] Taylor, R.L. (2014) ''FEAP - Finite Element Analysis Program'', Publisher: University of California, Berkeley, URL: http://www.ce.berkeley/feap.

[15] Sørensen, B.F. Jørgensen, K., Jacobsen, T.K., Østergaard, R.C. (2006) ''DCB-Specimen Loaded with Uneven Bending Moments'', Int J Fract. 141, 159-172.

[16] Rice, J. R. (1968) ''A Path Independent Integral and Approximate Analysis of Strain Concentration by Notches and Cracks'', J Appl Mech. 35, 379-386.