10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2019-162**
18-20 September 2019 - METU, Ankara TURKEY

SIMULATION OF INTERLAMINAR FAILURE IN CURVED UNIDIRECTIONAL LAMINATES UNDER MOMENT/AXIAL COMBINED LOADING

Tamer Tahir ATA[1]
Turkish Aerospace Industries, Inc
Ankara, Turkey

Demirkan COKER [2]
Middle East Technical University
Ankara, Turkey

ABSTRACT

In this study, interlaminar failure in unidirectional curved CFRP composite laminates is investigated numerically. The analyses are based on the experimental studies conducted by Tasdemir [Tasdemir, 2018]. Two dimensional finite element analyses of the unidirectional curved CFRP composite laminates under quasi-static loading are performed using ABAQUS / Explicit in conjunction with element-based cohesive behavior. Cohesive zone modelling is widely used in literature to predict delamination ([Uyar, 2014], [Gozluklu, 2015]). Failure load, location of damage initiation, the stiffness values before and after the failure are compared with experimental data. A major delamination is observed approximately at 35% of the thickness in experiments and 40% of the thickness in simulations. Finally, numerical results obtained from explicit analysis show good correlation with the experimental results.

INTRODUCTION

The usage of composite materials in primary and secondary structural elements of recent commercial aircrafts has been increasing rapidly due to their high specific stiffness and high specific strength. Compare to their high in-plane mechanical properties laminated composite materials inherently have low strength values in through the thickness region due lack of reinforcement in that direction. One of the widespread failure of laminated composites is delamination which can be clarified as layer separation or a crack between layers. Existance of delamination in a structure is sneaky, since the propagation of delamination may lead to ultimate failure.

Curved laminated composites suffer from delamination and matrix cracking due to development of significant tensile stresses in their curved region. With the intention of understanding failure initiation and propagation in curved region, computational investigation of curved laminated composites are performed under combined moment/axial loading. The main objectives of this study is to determine the failure load and failure patterns in simulations.

[1] Graduate Student in Aerospace Engineering Deparment, Email: ata.tamer@metu.edu.tr
[2] Associate Professor in Aerospace Engineering Deparment, Email: coker@metu.edu.tr

METHOD

In this section, the numerical approaches used in the simulations are explained in detail. The experiments conducted by Tasdemir [Tasdemir, 2018] are simulated by using finite element analysis in conjuction with cohesive zone modelling. The geometry of curved CFRP UD laminate is shown in Figure 1 (a). Inner radius (r_i) of the UD laminate is 8 mm. Arm lengths and width of the considered specimen are 66.36 mm and 25 mm, respectively. The loading fixture and installed specimen on it, are given in Figure 1 (b). The design of the experimental test fixture and installation of the specimen create moment/axial combined loading at the curved region. The experimental boundary conditions are idealized for finite element analysis as shown in Figure 1 (c).

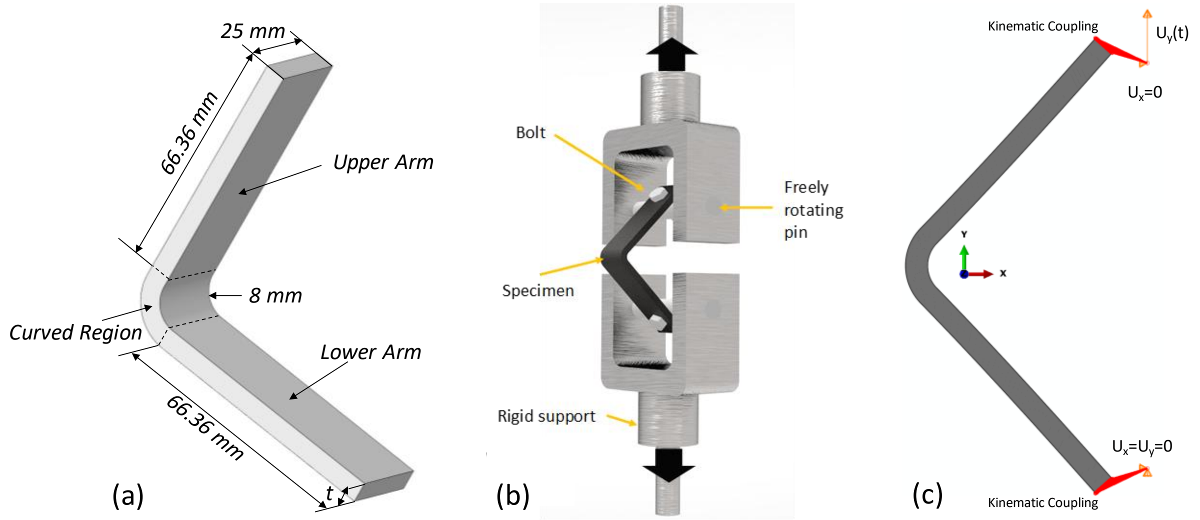

Figure 1: (a) Specimen geometry for the curved UD laminate, (b) Schematic of experimental configuration showing how the specimen is located in the fixture [Tasdemir , 2019], (c) Boundary conditions in 2D finite element model

In order to simulate the experimental configurations correctly, load and boundary conditions are imposed to the specimen using kinematic couplings which link the load introduction points to the specimen edge. Displacement control loading is applied to the specimen by using a smooth-step function which provides application of loading in a quasi-static manner.

The mechanical and interface properties of Hexply AS4/8552 UD Prepreg are provided in Table 1 and Table 2. All other values except interface strengths are directly taken from Camanho et al. [Camanho, 2009]. Interface strengths which denoted by t_I^0 and t_{II}^0 in Table 2 are obtained from experiments (Ata, 2019) conducted according according to ASTM Standard D6415 [ASTM, 2006] and ASTM Standard D2344 [ASTM, 2006]. The curve fit factor, η and the penalty stiffness value, K for mixed mode energy release rate calculation are also given in Table 2.

Table 1: Mechanical Properties of Hexply AS4/8552 UD Prepreg

Density	1580 kg/m^3
CPT	0.188 mm
Elastic Properties	E_{11}=135 GPa; E_{22}= E_{33}=9.6 GPa; G_{12}= G_{13}=5.3 GPa; G_{23}=3.4 GPa
	v_{12}= v_{13}=0.32; v_{23}=0.487
Strength (MPa)	X_T=2207; X_C=1531; Y_T=80.7; Y_C=199.8; S_L=114.5; S_T=128

Table 2: Interface Properties of Hexply AS4/8552 UD Prepreg

Interface Strength	$t_I^0 = 79.07\ MPa; t_{II}^0 = t_{III}^0 = 106.4\ MPa$
Fracture Toughness	$G_{I,C}$=0.28 N/mm; $G_{II,C} = G_{III,C}$ = 0.79 N/mm
B-K Criterion Constant (η)	1.45
Interface Stiffness	K=2.6×10^6 N/mm^3

The interface stiffness given in Table 2 is determined by using the below given closed-form expression derived by Turon et al. [Turon, 2005].

$$K = \frac{\alpha E_3}{t}$$

The two-dimensional finite element model of the considered UD laminate is generated in ABAQUS and explicit solver is employed due to the high dynamic nature of the damage in curved composite laminates. The model is composed of 30 layers of unidirectional layers discretized with 4-node linear, reduced integration elements (CPE4R) with enhanced hourglass control and 29 layers of interfaces discretized with 4-node two-dimensional cohesive elements (COH2D4). The main advange of using cohesive elements in simulation of delamination other than VCCT (Virtual Crack Closure Technique) is the capability of simulating damage initiation.

The mesh density at the curved region of the two-dimensional finite element model and details on interface modelling are shown in Figure 2. The width of the elements at the mid-line is w_e=94 µm and the height of the elements varies between 84 µm and 143 µm due to the curvature. The thickness of the cohesive elements at each layer interfaces is 0.001 mm.

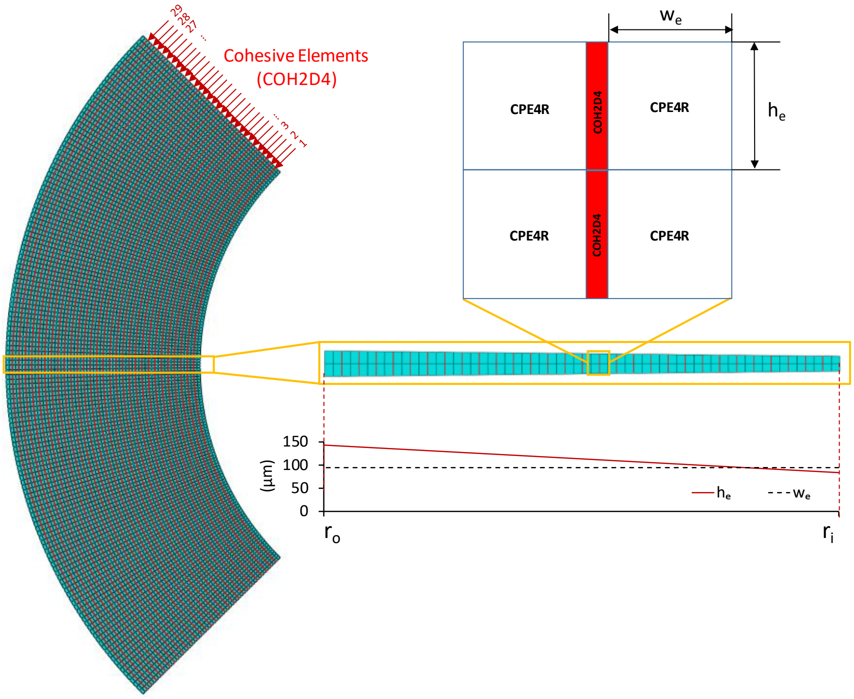

Figure 2: Mesh density at the curved region of the two-dimensional finite element model and details on interface modelling

The two-dimensional finite element model includes 52200 CPE4R and 25230 COH2D4 elements. The total number of element is 77430 and the total number of node is 78392 which corresponds to 156966 number of degrees of freedom. The stable time increment is calculated by ABAQUS as 1.472×10^{-10} s. The analysis was carried out on a high-performance cluster consisting of 72 CPU cores. A single simulation takes more than 33 h.

Quadratic nominal stress criterion is used to simulate mixed-mode delamination initiation. The Macaulay bracket used in the first term of the expression signifies that pure compressive stresses do not initiate damage.

$$\left\{\frac{\langle t_I \rangle}{t_I^0}\right\}^2 + \left\{\frac{t_{II}}{t_{II}^0}\right\}^2 + \left\{\frac{\langle t_I \rangle}{t_{III}^0}\right\}^2 = 1$$

Mixed-mode damage propagation is considered by using Benzeggagh and Kenane (1996) criterion:

$$G_{equiv,C} = G_{IC} + (G_{IIC} - G_{IC}) + \left(\frac{G_{II} + G_{III}}{G_I + G_{II} + G_{III}}\right)^{\eta}$$

where G_i,i is the work done by the pure mode traction t_i on the corresponding separation δ_i.

RESULTS AND DISCUSSION

The load-displacement response of the 2D simulation (red) of the curved unidirectional CFRP laminate under quasi-static moment-axial combined loading is compared with the experimental results (blue and orange) in Figure 3. The load-displacement behavior is linear elastic until sudden dynamic failure which occurs due to a single main delamination that extends to both arms from the curved region. The stiffness in elastic region is calculated as 206 N/mm and 197 N/mm for simulations and experimental work, respectively. The peak load before crack initiation is 1380 N for 2D Plane strain FEA and average of three experiments' failure loads is 1333 N which is within the scatter of the experiments.

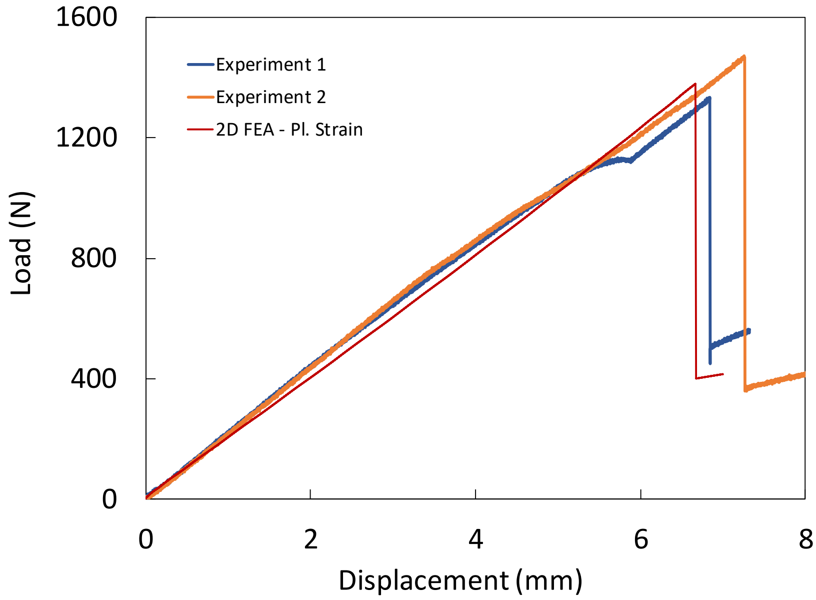

Figure 3: Load-displacement curves from 2D FEA and experiments of Tasdemir [Tasdemir, 2018]

Radial, tangential and shear stress contours at the curved region 0.5 µs before delamination initiation are shown in Figure 4. The maximum radial stress (79.08 MPa) location is observed to be the 12th interface which corresponds to 40% of the thickness from the inner radius (r_i). Due to the bending behavior of the applied load which tries to unfold the curved region, the tangential stresses at the inner radius region are tensile and the tangential stresses at the outer radius region are compressive. The shear stresses at the mid-line of the curved region are observed to be zero or close to zero. The shear stress values are increasing towards the arms.

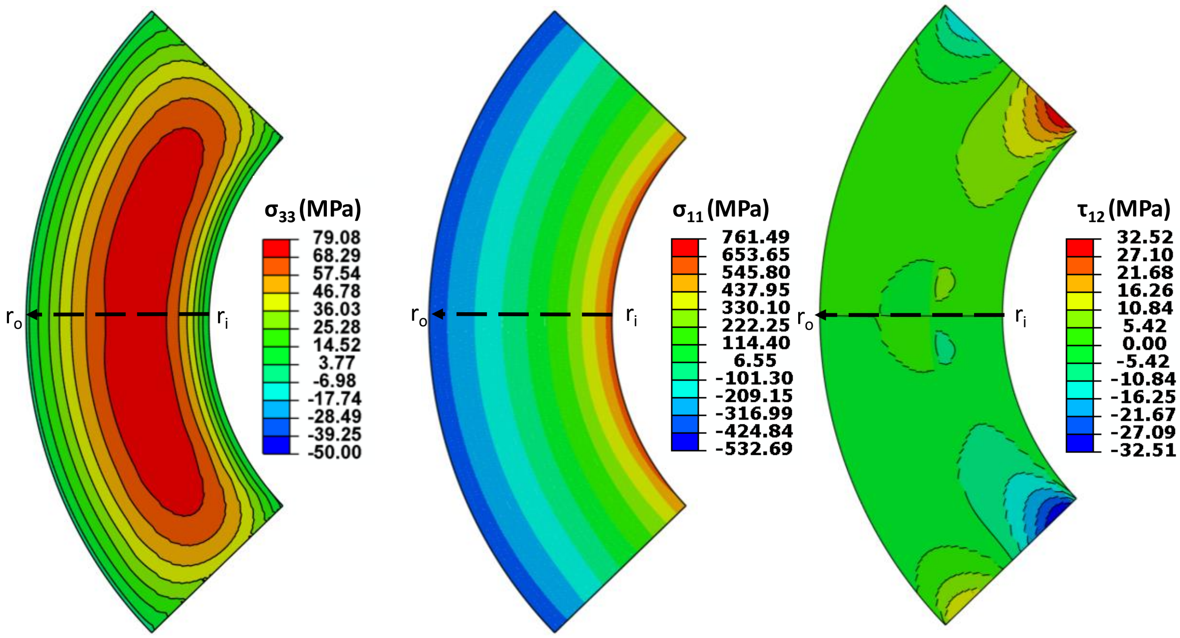

Figure 4: Radial, tangential and shear stress contours at the curved region 0.5 µs before delamination initiation

Radial and tangential stress values along the defined path from the inner radius to outer radius on the mid-line of the UD laminate are extracted from 2D finite element analysis and compared with the calculated analytical stress values according to analytical solution [Lekhnitskii, 1968]. Comparison graphs are only given for radial and tangential components, since shear stresses at the mid-line of the curve region are zero as shown in *Figure 4*. A slight difference is observed between FEA and analytical solution.

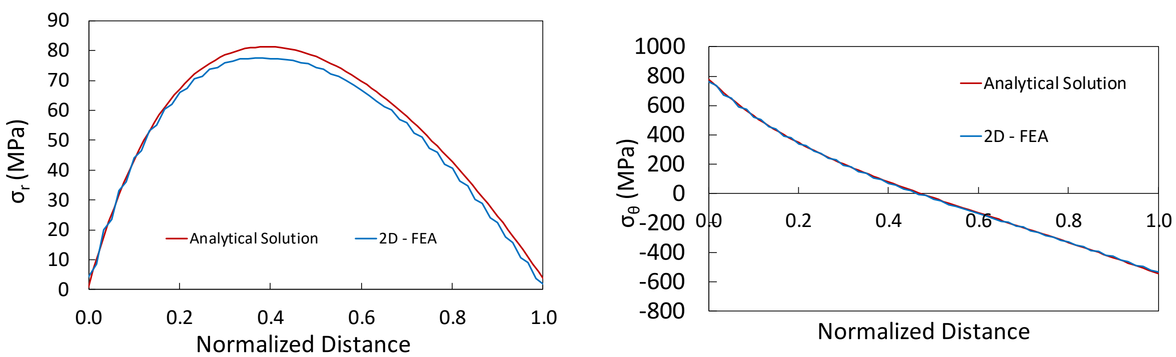

Figure 5: Comparison of 2D FEA and analytical solution for radial and tangential stresses along the defined path through the thickness at the mid-line of the curve.

Radial stress distribution before and after delamination initiation is given below in Figure 6. The time instant at which the first cohesive element completely degrades is set as t_0. Delamination initiates exactly at the maximum radial stress location which is the 12th interface of the 30 layer specimen. Hence failure location is 40% of the thickness from inner radius which is consistent with experimental observation (35%) and analytical calculations.

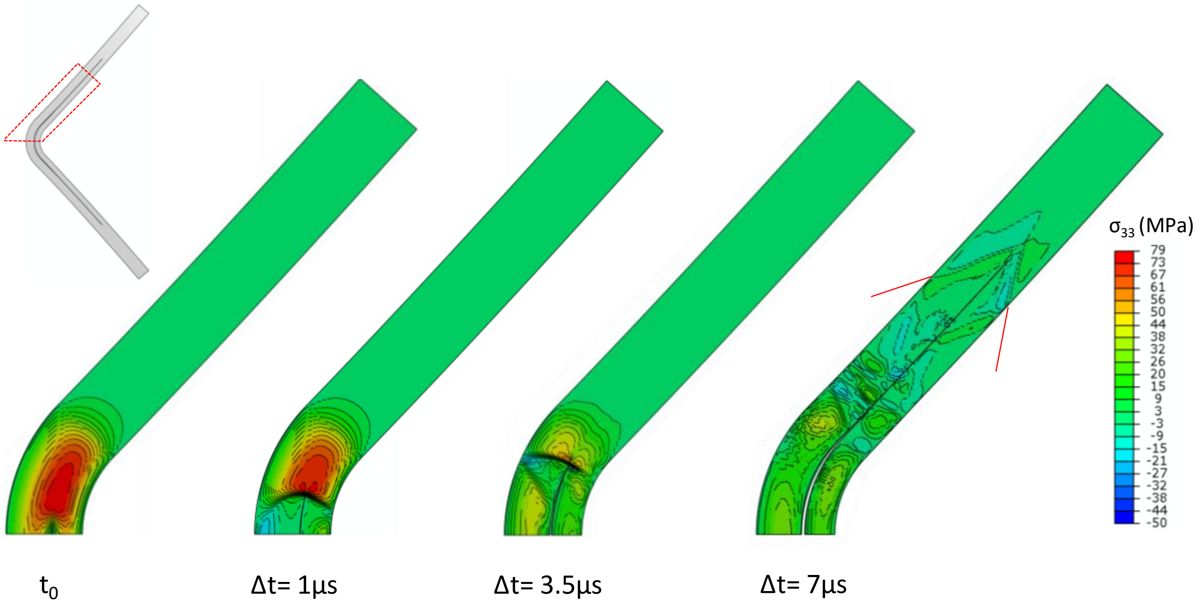

Figure 6: Stress distributions before and after delamination initiation

The location of delamination on fully deformed specimen from experiment [Tasdemir, 2018] and 2D finite element analysis are shown in Figure 7. This qualitative comparison reveals that the the delamination location compares well with the experiments.

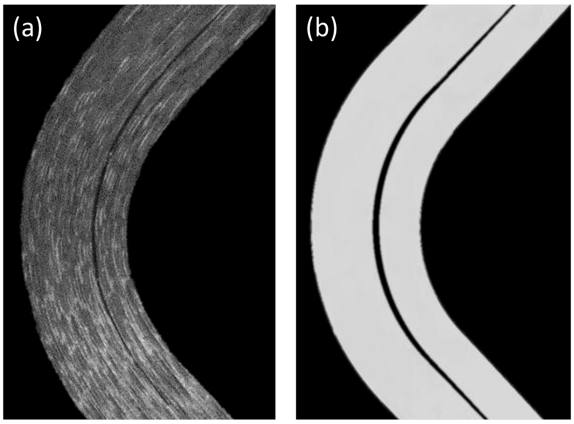

Figure 7: Delamination location on fully deformed specimen from (a) experiment [Tasdemir, 2018] and (b) 2D finite element analysis

CONCLUSION

- Finite element modelling with cohesive elements correctly simulated delamination initiation and propagation in unidirectional laminates.
- A major delamination is observed approximately at 35% of the thickness in experiments and 40% of the thickness in simulations.
- Numerical results obtained from explicit analysis show good correlation with the experimental results in terms of failure load, location of damage initiation, the stiffness values before the failure.
- Evaluating the stress states at the curved and arm region reveals that the delamination initiates in Mode-I and as it propagates through the arm region Mode-II also becomes effective and mixed-mode delamination propagates to the end of the specimen.

References

ASTM. ASTM Standard D6415. *Standard Test Method for Measuring the Curved Beam Strength of a Fiber-Reinforced. West Conshohocken PA:* 2006.

ASTM. ASTM Standard D2344. *Standard Test Method for Short-Beam Strength of Polymer Matrix Composite Materials*. *West Conshohocken PA:* 2006.

Ata, T. T. (2019). 2D and 3D Finite Element Analyses of Dynamic Delamination in Curved CFRP Laminates. MSc thesis, Middle East Tech Univ.

Uyar I., Gozluklu B., Coker D. (2014) *Dynamic Delamination in Curved Composite Laminates Under Quasi-Static Loading*. vol. 524, 2014.

Gozluklu B., Uyar I., Coker D. (2015) *Intersonic Delamination in Curved Thick Composite Laminates under Quasi-Static Loading, Mechanics of Materials*, vol. 80, pp. 163-182, 2015.

Lekhnitskii S. G. (1968) Anisotropic plates. Amsterdam: Gordon & Breach; 1968.

Lopes C. S., Camanho P. P., Gürdal Z., Maimí P. and González E. V. (2009) *Low-Velocity Impact Damage on Dispersed Stacking Sequence Laminates.* Part II: Numerical simulations, Compos. Sci. Technol., vol. 69, no. 7–8, pp. 937–947, 2009.

Tasdemir, B. (2018) *Fatigue and Static Behavior of Curved Composite Laminates.* MSc Thesis, Middle East Technical University,2018.

Tasdemir B. and Coker D., (2019) Comparison of Damage Mechanisms in Curved Composite Laminates Under Static and Fatigue Loading, Composite Structures, Volume 213, 2019, p: 190-203.

Turon A., Dávila C. G., Camanho P. P. and Costa J. (2005) *An Engineering Solution for using Coarse Meshes in the Simulation of Delamination with Cohesive Zone Models.* NASA/TM-2005-213547, no. March. p: 1–26, 2005.