

Industrial Training Report

Ana Salas Ordóñez

June 20, 2017

Mesh generator in Matlab for numerical models in Pam-Crash and Abaqus.

1 Introduction

In this report, the tasks carried out in the company IDIADA Automotive Technology S.A., corresponding to the Industrial Training of the Master on Numerical Methods in Engineering, are presented. In order to learn and understand the tools used in the company for the simulation of crash events, a mesh generator with MATLAB has been made for numerical models in PAM-CRASH and ABAQUS. The models developed for PAM-CRASH will be used in the master thesis *Damage and failure modelling of composite materials for the automotive industry*.

The software used as preprocessor and postprocessor are ANSA and META, respectively. PAM-CRASH and ABAQUS are solvers based on the FEM algorithm used for crash simulation and the design of occupant safety systems, primarily in the automotive industry. The idea of the mesh generator is to obtain directly the file *.txt* containing the geometry, boundary conditions and material cards necessary to run the solver. For this, numerical models of different delamination tests (DCB, C-ELS, MMB) in a composite material, used to characterize the interphase properties, are presented.

2 Numerical models

The scheme of the generic specimen used for the delamination tests is shown in Figure 1. The values of the parameters defining the geometry will be detailed for each test. For these tests, an unidirectional carbon-reinforced composite conformed with carbon fibres G1157.6K is used. The fibres are embedded in an Epoxy matrix with reference “Binder 180°C Epoxy Power”. These tests are performed over pre-cracked specimens with this material. Moreover, the configuration of the specimen is: 12 plies oriented at 0° with a total thickness of 3.144 mm.



Figure 1: Geometric definition of the generic specimen.

2.1 Double-Cantilever Beam (DCB)

For the numerical model of Double-Cantilever Beam (DCB) test, the values of the parameters that define the geometry of the specimen (Figure 1) are the following:

- Length of the specimen ($2L$): 250 mm.
- Width of the specimen (b): 25 mm.
- Initial crack length (a_0): 25 mm.

The boundary conditions for the numerical simulation consist of two imposed displacements in opposite z -directions at the tip of the specimen ($x = 2L$) and all nodes clamped in $x = 0$.

2.2 Calibrated End-Loaded Split (C-ELS) test

For the numerical model of Calibrated End-Loaded Split (C-ELS) test, the values of the parameters that define the geometry of the specimen (Figure 1) are the following:

- Length of the specimen ($2L$): 105 mm.
- Width of the specimen (b): 25 mm.
- Initial crack length (a_0): 57 mm.

The boundary conditions for the numerical simulation consist of one imposed displacement in the z-direction at the tip of the specimen ($x = 2L$) and all nodes clamped in $x = 0$.

2.3 Mixed-Mode Bending test

For the numerical model of Mixed-Mode Bending test, the values of the parameters that define the geometry of the specimen (Figure 1) are the following:

- Length of the specimen ($2L$): 100 mm.
- Width of the specimen (b): 25 mm.
- Initial crack length (a_0): 25 mm.

The boundary conditions for the numerical simulation consist of one imposed displacement in the z-direction at the centre of the loading arm, nodes at $x = 0$ with translation fixed in y and z directions and rotation fixed in z-direction, nodes at $x = 100$ with translation fixed in x,y,z-directions and rotation in z-direction

3 Results

In this section the mesh of the numerical models previously described are presented. In addition, the corresponding *.txt* files to run the solver have been obtained, but they are not presented in this report due to confidentiality issues.

In order to simulate the delamination mechanism, the behaviour of the interface between the central plies of the laminate must be simulated. Numerical models are performed over two plies linked by cohesive elements (C0H2D4), in ABAQUS, and tie elements, in PAM-CRASH. The material labelled as 303 in PAM-CRASH is chosen to define the mechanical properties used in the two tied interfaces. Since it is the one that simulates

adhesive unions, is the best option to simulate the interface between plies.

On the other hand, in ABAQUS each sub-laminate of the composite material is modelled using two-dimensional 4-node bilinear solid elements (CPE4R), while in PAM-CRASH is modelled using 4-node shell elements and the material labelled as 131, which corresponds to multilayered and multi-material shell models in which the layers can consist of material modelled as elastic damaging fiber-matrix composite. Ply properties are selected from the composite ply card. In ABAQUS only the numerical model of DCB test is made. For each test, a mesh with elements of 5 mm is used.

The loading arm in the MMB test (Figure 5) is modelled as a rigid body, with a null material labelled as 100. A null material is used when the internal forces and deformation of a surface is not of interest, as it is in the case of the loading arm. In addition, a contact algorithm is included in order to avoid interpenetration between the different plies when a tie element is eliminated. For this, the contact labelled as “36”, which corresponds to a “Self-Impacting Node-to-segment with edge treatment”, is chosen. However, no contact algorithm is used for the interaction between the loading arm and the plies, since this part is only used to apply the load in the centre of the arm in order to behave as a lever and therefore, it does not matter if it penetrates the plies.

The mesh obtained in ANSA with the .txt file generated in MATLAB, is represented in Figure 2(b). On the other hand, in Figures 2(a) and 3 can be observed the differences of modelling the plies and the cohesive zone with both FEM solvers, ABAQUS and PAM-CRASH, respectively. For the models of PAM-CRASH, the tie elements are represented as red lines in MATLAB. The results obtained with the simulation of these tests are not presented in this report, but in the master thesis *Damage and failure modelling of composite materials for the automotive industry*.

3.1 Double-Cantilever Beam (DCB)

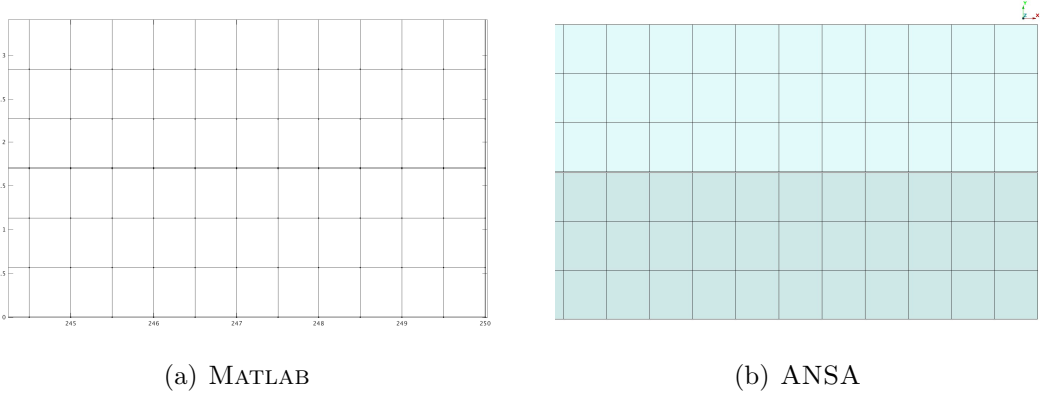


Figure 2: Numerical model for Double-Cantilever Beam (DCB) test in ABAQUS.

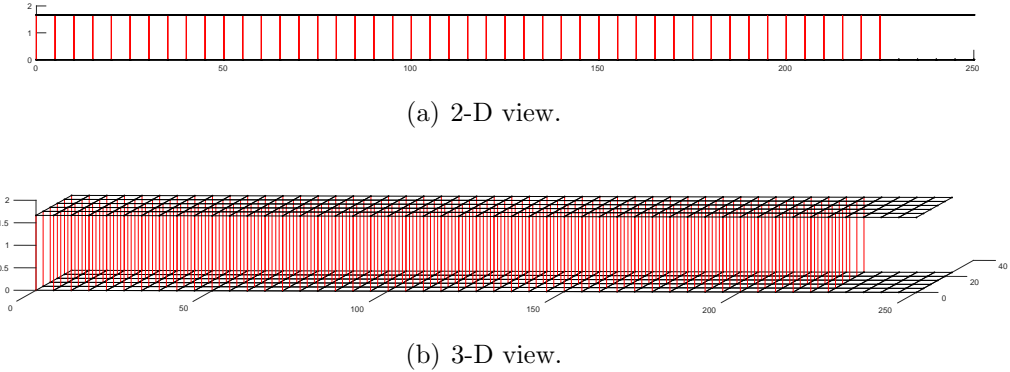


Figure 3: Numerical model for Double-Cantilever Beam (DCB) tests in PAM-CRASH.

3.2 Calibrated End-Loaded Split (C-ELS) test

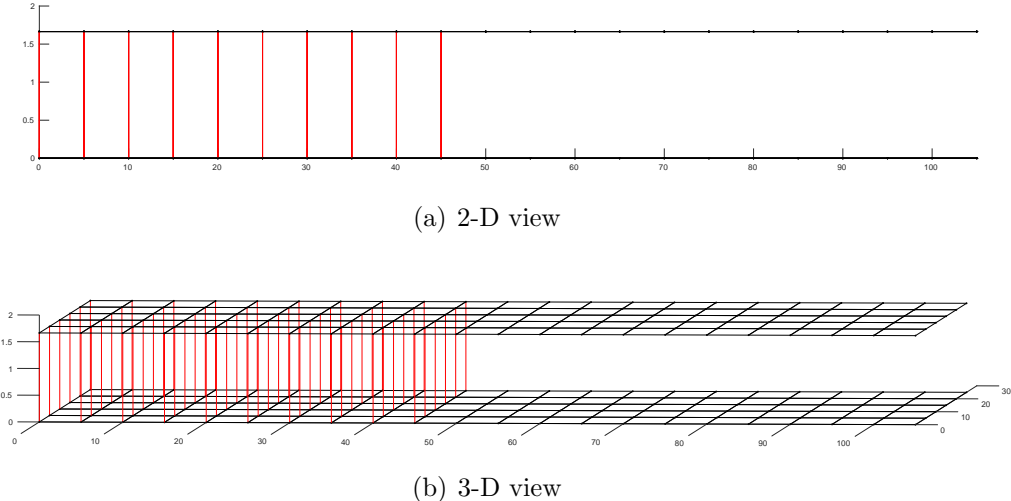


Figure 4: Numerical model for Calibrated End-Loaded Split (C-ELS) tests in PAM-CRASH.

3.3 Mixed-Mode Bending test

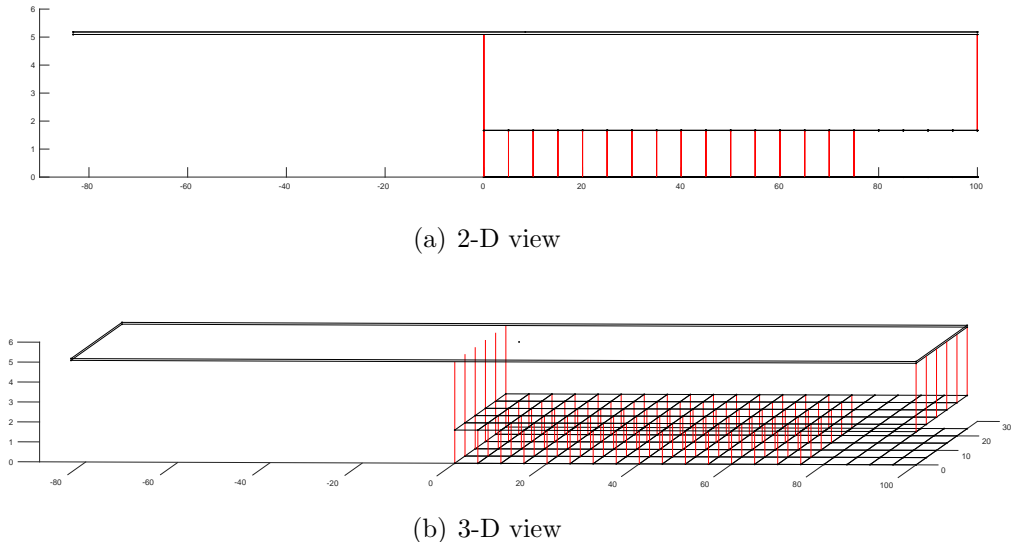


Figure 5: Mixed-Mode Bending (MMB) test configuration in PAM-CRASH.