16th International Conference on Composite Structures
ICCS 16
A. J. M. Ferreira (Editor)
© FEUP, Porto, 2011

INTERFACE ELEMENT FOR DELAMINATION SIMULATION A GOOD USAGE FOR ACURACY AND PERFORMANCES

Michaël Bruyneel*, Jean-Pierre Delsemme*, Frédéric Duboeuf*, Philippe Jetteur*

* SAMTECH Headquarters
LIEGE science park - Rue des Chasseurs-Ardennais, 8
B-4031 Liège (Angleur), BELGIUM

Key words: Composite FE modeling, cohesive, interface element, delamination.

Summary. This paper deals with the use of interface element for the simulation of crack propagation. The questions: "how to choose mesh size, material properties and model parameters in order to get a correct result in a reasonable time" will be discussed. An industrial test case with skin-stringer separation will also be presented.

1 ABSTRACT

Interface elements are nowadays widely used to simulate crack propagation in composite materials. They can be simply introduced between shell or solid elements. The element thickness is not generally deduced from node coordinates but is assigned to the element as a property defined by the user. Several material laws can be used to simulate the local behavior. The three available in *SAMCEF* are: exponential, bi-triangular and polynomial (LMT Cachan[1]) It is well known that the area under the curve is the energy-release rate of the interface. For propagation in mode 1, this parameter can be determined by a standard DCB test. The problem is that the material laws have one, two or more additional parameters. They can't be assessed by an experimental test; it is the user's responsibility to choose the best values in order to get a correct result but in a reasonable computation time.

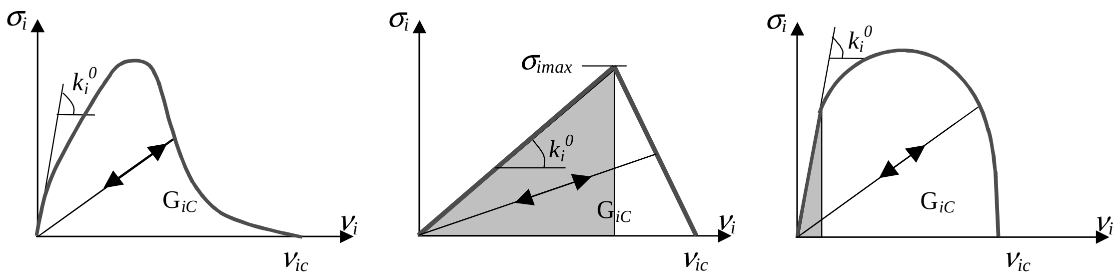

Figure 1: The three cohesive laws available in SAMCEF

The initial stiffness (k_i^0) of the local law is directly or indirectly related to the model parameters. This stiffness influences the stress distribution near the crack tips. The stiffer it is the faster is the stress variation. This variation of stress must be made coherent with the mesh size. And of course the mesh size influences the computation time.

The crack propagation is, by nature, a hard discontinuity in the behaviour of the structure. From computational point of view it makes non-linear equations hard to solve with the

Newton's method. In order to smooth nonlinearities, it is common to add a delay effect[2] to the law. It has been shown that this effect eases the convergence but also makes the results less sensitive to the mesh size.

After a brief overview of the material laws mentioned above, the paper will present an extensive analysis of the influence of various parameters of the simulation based on a DCB test (mesh size, initial stiffness, maximum stress, tangent matrix option, integration scheme (Gauss or Lobatto)). Accuracy of the solution will be discussed but a special attention will be paid to the cost of the computation in terms of Newton's iterations.

As a conclusion, the paper will also present the simulation of a CFRP stiffened panel in compression up to collapse. The simulation of panel buckling including skin-stringer separation will be shown. By comparison to experimental result, the influence of various simulation parameters will be presented.

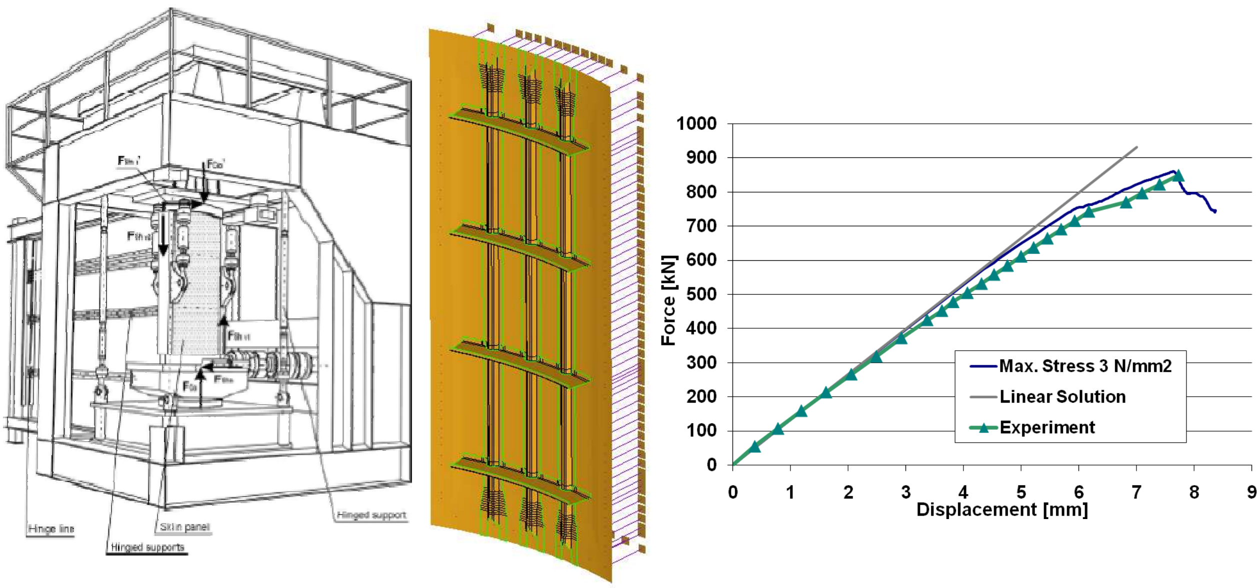

Figure 2: Panel buckling simulation with skin-stringer separation.

REFERENCES

[1] O. Allix; "The damage mesomodel for laminates interface identification" LMT Chachan

[2] O. Allix, "Delay damage modelling for fracture prediction of laminated composite under dynamic loading" LMT Chachan

[3] J-P. De1semme, C. Brauner, M. Bruyneel, P. Jetteur "Simulation of Stiffened Composite Panel Buckling considering Ply Degradation and Skin-Stringer Separation" SAMPE Europe SEICO 10, Int. Conf. Paris.