BEHAVIOUR OF SOLID LIKE SHELL ELEMENTS IN SIMULATION OF COMPOSITES IN AIRCRAFT STRUCTURES

Peter Linde*, Henk De Boer†

* Airbus, Kreetslag 10 DE-21129 Hamburg, Germany
e-mail: peter.linde@airbus.com, web page: http://www.airbus.com

† Advanced Lightweight Engineering Rotterdamseweg 145 NL-2628 AL Delft, The Netherlands
e-mail: deboer@ale.nl, web page: http://www.ale.nl

Key words: Composite structures, Modelling, Shell elements, Damage behaviour.

Abstract: In prediction of the structural behavior of airframe structures of carbon reinforced composites, nonlinear finite element models are often necessary, if the behavior is to be studied to final failure of the structure. For modeling e.g. test specimens larger than the smallest coupons, shell elements have been used in the finite element model to represent e.g. skin, stringers, or frames, since any more detailed modeling has become numerically to complex and expensive.

Advances in shell element technology has enabled quite realistic simulations with one classical shell element through the thickness, including layered material modeling so as to model each ply. However, intralaminar delamination cannot be well simulated with these elements. If a more detailed modeling was needed, with an element over the thickness of each ply, the only alternative until recently has been resorting to solid elements. However, due to their aspect ratio limitations, a very fine mesh was then usually necessary, resulting in an unacceptable numerical expense.

Numerical research in the late 1980s and 1990s led to the development of shell elements with thickness dimensions and thickness stress; so called “solid like shell” elements, [1, 2, 3]. They remained research activities until some commercial finite element vendors introduced simple versions of them around 2004, amongst them ABAQUS and ANSYS. These elements can theoretically be stacked on top of each other, and thus model a laminate over the thickness, one element per ply, and use larger aspect ratios than solid element permit.

A major factor for their commercial introduction was requirements by mechanical industry to be able to readily build a mesh of shell elements, based on 3D CAD files, rather than refined analysis of composites. The application of these elements in detailed damage analysis for composites in aircraft industry has not been fast. There are several reasons for this: the
elements employ “reduced” integration, and the thickness behavior is hardly rudimentary described.

In this paper a solid like shell element, SLS-element, with a clear element formulation is implemented in a user element sub routine in ABAQUS and evaluated in terms of its behavior in carbon composite simulations.

The element, originating from research at the university of Stuttgart in the 1990s, has intermediate nodes over the thickness, and a through the thickness stress. This is not the case with corresponding ABAQUS-“continuous shell” element, SC8R, which is used in this study as reference [4].

An accurate through the thickness stress should be important in damage simulations of composites laminates. This is due to the fact that no damage model that includes out of plane stress or strain, will behave “better” than the accuracy of stress, that it obtains in this direction from the element in which is implemented, permits.

In this paper, basic behavior of the SLS-element, compared with the SC8R-element of ABAQUS is performed, at first with only one element over the skin thickness, to test rudimentary behavior. Then its use in a stiffened panel model is assessed, whereby numerical efficiency and convergence is compared. Finally the behavior in a refined modeling with one element per ply of a coupon specimen is studied, combined with material damage behaviour, again with the SC8R element as reference.

Finally, conclusions and recommendations for future research are provided.

REFERENCES