VALIDATION OF LAMINATED COMPOSITE SHELL ELEMENTS TO COUPON TEST RESULTS USING MSC NASTRAN, ABAQUS & LS-DYNA

Dr Mark Arnold & Andy Ngai

Penso

ABSTRACT

Laminated composite shell elements available within a number of industry-standard implicit and explicit finite element analysis solvers were validated against coupon test results to understand current capabilities for predicting macroscopic structural behaviour of continuous fibre reinforced composite materials. This simulation was based upon methods developed whilst engineering lightweight composite structures for aerospace and automotive customers at Penso, including the monocoque chassis for the Jaguar C-X75 prototype.

FE models comprising laminated composite shell elements were generated using the ANSA pre-processor to represent a number of rectangular test specimen geometries. Linear and non-linear stress-strain response orthotropic ply material models were then defined for a number of bi-directional, 'woven fabric' carbon fibre-epoxy resin prepregs based upon mechanical properties in warp and weft directions. The strength and modulus constants were characterised by the material supplier from static tensile, compressive and in-plane shear coupon tests carried out according to common ASTM standards on specimens cut from autoclave laminates.

Static analysis using MSC Nastran, Abaqus and LS-DYNA was undertaken to simulate the same tensile, compressive and in-plane shear tests used to characterise the material models, as well as interlaminar shear and flexural tests additionally performed by the material supplier. Failure loads, displacements and stresses in local (ply direction) coordinate system were extracted from analysis results using μETA post-processor and compared with test results to validate accuracy of the FE model. Comparisons were made between different failure criteria (interactive and non-interactive) and progressive damage models available within each solver, considering accuracy in predicting the failure load and mode of failure.

Sensitivity of analysis results to variations in modelling parameters and solution settings were quantified, including shell element formulation, material properties and representation of loading/constraint applied by test fixtures. Verification of the mathematical model was also considered by comparing the results predicted by each FEA solver, whilst element size was also varied to

quantify discretization error in the numerical solution. Finally, recommendations were proposed to ensure a robust correlation.

SUGGESTED THEMES

Composite, material, validation, Nastran, Abaqus, LS-DYNA.