Development and validation of a composite fastened joint model using advanced measurement techniques

DEVELOPMENT AND VALIDATION OF A COMPOSITE FASTENED JOINT MODEL USING ADVANCED MEASUREMENT TECHNIQUES

Nikolaos Perogamvros, George Lampeas*

Laboratory of Technology and Strength of Materials
Department of Mechanical Engineering and Aeronautics
University of Patras, 26500 Rion, Greece
nperogam@mech.upatras.gr, labeas@mech.upatras.gr

ABSTRACT

During the last decades, the application of fibre-reinforced composite materials in the design of advanced automotive and other transportation structures is growing due to their inherent advantages, such as high specific stiffness and strength, long fatigue life and superior corrosion resistance. However, the potential implementation of such complex and lightweight material systems imposes additional restrictions, specifically when different composite or hybrid composite/metallic parts are fastened together in order to construct the complete structure. Another limitation of that type of composite constructions is the poor resistance in the out-of-plane loads, leading to a failure of the matrix-rich interlaminar regions. Such an interlaminar failure (or delamination) is the dominant failure mode, particularly when the composite structures are loaded transversely at impact rates. Therefore, the design of the joints can have a large influence over the structural integrity and the load carrying capability of the overall structure. Due to these reasons, the mechanical response of the hybrid composite/metalic fastened parts, as well as the three-dimensional (3D) stress and strain fields which occur near the vicinity of the fastened areas and lead into different macroscopic failure modes (e.g. pull-through, bearing, net tension) need to be fully understood and examined.

In the present work, a representative element of an automotive stiffened hybrid composite fastened joint is examined through explicit finite element analysis and dynamic experimental testing.

A novel simulation methodology in the frame of FE, which is based on the stacked shell approach for composite laminates, is used for the modelling of the fastened coupons. In the stacked shell approach, one element through the thickness of the composite material is used to represent one or a small set of plies, while cohesive interfaces are introduced between the elements. The specific methodology offers acceptable computational cost, combined to accurate out-of-plane damage prediction (delamination), and can be potentially combined with Progressive Damage Modelling (PDM) approaches. The LS-DYNA non linear FE code is used for the coupon simulation while different representation capabilities of the cohesive zone modelling are also examined.

The FE model was validated via experimental tests supported by advanced full field optical measurements of displacement and strain histories, using a 3D Digital Image Correlation optical system Q-450 by Dantec Dynamics. The experimental results were successfully compared against the respective numerical data using conventional and advanced validation methodologies.

Acknowledgement

The research leading to these results has received funding from the European Union Seventh Framework Programme (FP7/2007-2013) under grant agreement n° 314744.