Comparison of 2D-shell and 3D-solid elements formulations for modelling coupons at dome area of composite pressure vessels

M. Bruyneel[$,*], A. Rajaneesh[$], F. Strepenne[$], M. Haeseleer[$]
[$] GDTech, Alleur, Belgium
[*] University of Liège, Belgium

T. Watanabe, Y. Urushiyama, Y. Tsuchiyama
Honda R&D Co., Ltd. Innovative Research Excellence, Power Unit and Energy, Tochigi, Japan

ABSTRACT

This paper describes a methodology developed to study and model the material properties in the dome and in the cylindrical area of composite overwrapped pressure vessels (COPV). For that purpose, a COPV is produced on a hexagonal-shaped so-called "soccer ball" mandrel, which has cleverly-designed flat faces in the dome and cylindrical regions, tests on coupons are performed, and a specific FEM modelling methodology is developed.

Firstly, coupons with specific dimensions and stacking sequences are extracted from the cylindrical region of the COPV and are tested. The intra-laminar material parameters of an advanced damage model of an orthotropic ply [1] are then identified, neglecting the interweaving of the tapes deposited during filament winding (see Figure 1).

Secondly, coupons in the dome area are defined in the longitudinal and circumferential directions. These coupons are flat on one side (the one in contact with the mandrel), and non-flat on the other side since they present a (complex) variable thickness pattern. These coupons are tested in four-points bending.

The first step of the modelling process consists in reproducing the deposition of successive tapes on the mandrel. Based on that information, models of the coupons are developed.

In this paper, we compare the solutions obtained with two modelling approaches. The first one is based on using multilayer shell elements, and the second one relies on 3D solid elements models inflated based on the shell models. In this last approach, each layer is represented in 3D; we then come up with a 3D model with variable thickness that represents the interlacements of the tapes resulting from the filament winding process.

The material model with the parameters previously identified is then assigned to the shell and 3D models of the coupons in longitudinal and circumferential directions. Four-point bending physical tests are conducted on the coupons in longitudinal and circumferential directions, and the corresponding FEM simulations are performed. Figure 1 compares tests and simulation results for the longitudinal coupon at the dome area with a model based on 3D solid elements. A very good agreement is obtained between simulations and tests in terms of stiffness and strength (maximum force at failure). In the paper, we also explain how the 3D-solid element model is developed, and we study the influence on the numerical results of the mesh size and the tapes boundary representation, and of the tape thickness representation through the coupon thickness.

[1] Rajaneesh A., Bruyneel M. (2023). Low-velocity impact and compression after impact modeling of composites using modified mesoscale model. Composite Structures, 311, 116821.

[2] Bruyneel M., Rajaneesh A., Watanabe T., Urushiyama Y., Tsuchiyama Y. (2023). Validation of composite pressure vessels modelling methodology using 2D-shell elements at coupon level. ECCOMAS Thematic Conference on Mechanical response of Composites, September 12-14, Trapani, Sicily, Italy.

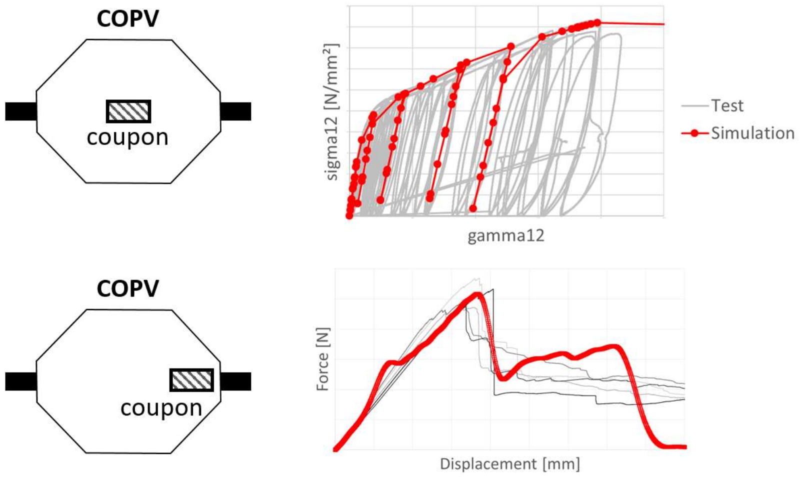

Figure 1. Results of the material parameters identification process, and tests/simulation comparison for 4-points bending on a coupon extracted from the dome area