10th International Conference on Composite Science and Technology
ICCST/10
A. L. Araújo, J. R. Correia, C. M. Mota Soares, et al. (Editors)

Abstract ID-221

A NUMERICAL SIMULATION OF THE STRENGTH AND STIFFNESS OF COMPOSITE STRUCTURES

Daiva Zeleniakiene, Paulius Griskevicius, Vitalis Leisis

Kaunas University of Technology, Lithuania
daiva.zeleniakiene@ktu.lt, paulius.griskevicius@ktu.lt, vitalis.leisis@ktu.lt

Keywords: sandwich composite, hollow cylindrical, structure, stress, stiffness, finite element modelling

Summary: The object of the investigation was a cylindrical structure made of sandwich composite with facesheets from filament wound glass fibre and polyvinylester resin and a core from recycled paper hexagonal honeycomb impregnated with polyvinylester resin.
The aims of the study were: to verify experimentally the numerical models of materials and sandwich structure for the investigation of the strength and stiffness of hollow cylindrical structures; using verified numerical model to determine the optimal geometrical parameters which ensures the stiffness and strength properties with the largest efficiency of fibre reinforced plastic volume usage of cylindrical structures.
Using experimentally obtained data of materials properties and the finite element (FE) code LS-DYNA the numerical FE model of sandwich structure was designed. To verify the FE model three different tests were performed. In the first step the verification of the numerical material model by simulating the uniaxial tensile test of facesheets has been performed. The numerical results of elastic behaviour showed good agreement with experimental stress – strain curve.
The verification of sandwich structure model was performed by simulating three points bending and the compression of hollow cylinder tests. In both the numerical and experimental investigations the force and deflection were measured and compared. During the ring compression tests the core failure was followed after debonding as in three point bending test. Debonded zones of tested specimens coincided with the FE simulation results.
The verified model allowed investigation of the stiffness and strength of cylindrical structures and determination of the optimal geometrical parameters.