World Academy of Science, Engineering and Technology
International Journal of Materials and Textile Engineering
Vol:14, No:09, 2020

Finite Element Modelling of a 3D Woven Composite for Automotive Applications

Authors : Ahmad R. Zamani, Luigi Sanguigno, Angelo R. Maligno

Abstract : A 3D woven composite, designed for automotive applications, is studied using Abaqus Finite Element (FE) software suite. Python scripts were developed to build FE models of the woven composite in Complete Abaqus Environment (CAE). They can read TexGen or WiseTex files and automatically generate consistent meshes of the fabric and the matrix. A user menu is provided to help define parameters for the FE models, such as type and size of the elements in fabric and matrix as well as the type of matrix-fabric interaction. Node-to-node constraints were imposed to guarantee periodicity of the deformed shapes at the boundaries of the representative volume element of the composite. Tensile loads in three axes and biaxial loads in x-y directions have been applied at different Fibre Volume Fractions (FVFs). A simple damage model was implemented via an Abaqus user material (UMAT) subroutine. Existing tools for homogenization were also used, including voxel mesh generation from TexGen as well as Abaqus Micromechanics plugin. Linear relations between homogenised elastic properties and the FVFs are given. The FE models of composite exhibited balanced behaviour with respect to warp and weft directions in terms of both stiffness and strength.

Keywords : 3D woven composite (3DWC), meso-scale finite element model, homogenisation of elastic material properties, Abaqus Python scripting
Conference Title : ICACT 2020 : International Conference on Automotive Composites and Textiles
Conference Location : Rome, Italy
Conference Dates : September 17-18, 2020