A finite element model of crimped fibril reinforced composite to predict the mechanical response subjected to tensile load

Li Liu[1], *Dean Hu[1], and Xu Han[1]

1State Key Laboratory of Advanced Design and Manufacturing for Vehicle Body, Hunan University, Changsha 410082, P. R. China

Presenting author: liugyli@hnu.edu.cn

*Corresponding author: hudean@hnu.edu.cn

Abstract

This paper simulates the mechanical responses caused by the geometric effect of crimped fibers subjected to tensile load. A periodic unit cell, a sinusoidal waviness fiber explicitly embedded into the matrix, is developed using the commercial codes ABAQUS. The matrix is assumed to be linear elastic and isotropic, whose Young modulus (E_m) and Poisson ratio (ν_m) are constants. The fiber is modeled as transversely isotropic elastic material and the modulus variation versus the fiber strain levels is considered; the strain-dependent fibril Young's modulus along the fiber-chord direction is employed to the user subroutine USDFLD. In addition, considering the fiber-matrix interaction effect, the cohesive interface element with zero thickness is adopted using the bilinear constitutive relationship. Then, the periodic boundary condition is applied to the representative unit cell. Finally, the fibers with different shape parameters (a/λ, ω and r/λ) in the crimped reinforced composites are investigated to study the corresponding mechanical responses. The accuracy of the proposed model is verified by comparing with the existing findings. The results indicate that the longitudinal Young's modulus decreases with the increasing of the fiber waviness a/λ. However, as the shape parameters of the ω and r/λ increase, the longitudinal Young's modulus increases. The predictions of the proposed model are in very good agreements with the results obtained from nonlinear spring models.

Keywords: Composite material, crimped fiber, periodic unit cell, periodic boundary, geometric effect, mechanical response