

Finite element modelling for predicting the puncture responses in papayas

ABSTRACT

This study aims to develop a finite element (FE) model to determine the mechanical responses of Exotica papayas during puncture loads. The FE model of the puncture-test was developed using the ANSYS 19.1 software. The proposed framework combined the finite element method and statistical procedure to validate the simulation with the experimental results. Assuming the elastic-plastic behaviour of papaya, the mechanical properties were measured through tensile test and compression test for both skin and flesh. The geometrical models include a quarter solid of papaya that was subjected to a puncture test with a 2 mm diameter flat-end stainless-steel probe inserted into the fruit tissues at 0.5 mm/s, 1 mm/s, 1.5 mm/s, 2 mm/s, and 2.5 mm/s. The FE results showed good agreement with the experimental data, indicating that the proposed approach was reliable. The FE model was best predicted the bioyield force with the highest relative error of 14.46%. In conclusion, this study contributes to the usage of FE methods for predicting the puncture responses of any perishable fruit and agricultural products.

Keyword: Papayas; Finite element method; Puncture mechanics; Yielding behaviour