

A Simple Efficient Finite Element for the Sandwich Plates Analysis

K. BELKAID, B. BOUBIR, N. BOUTASSETA, H. Aouaichia, N. Fergani, D. GAAGAIA

Abstract : In this work, a numerical simulation has been considered of a sandwich composite panel subjected to a mechanical uniaxial load in order to study the buckling behaviors. In this study, we used the finite element Abaqus software and a simple efficient finite element model available in the literature [1] in order to: 1- Describe the fundamental buckling modes of a sandwich composite panel. 2- Understand the buckling behavior on this type of structure. 3- Validate the reply of the developed element on buckling behavior for the sandwich composite structure with the finite element ABAQUS software.

Keywords : composite, sandwich, buckling modes, finite element