MATERIALS AND STRUCTURES SYMPOSIUM (C2)
Space Environmental Effects and Spacecraft Protection (6)

Author: Prof. Harijono Djojodihardjo
Universitas Al Azhar Indonesia, Indonesia, harijono@djojodihardjo.com

GENERIC STUDY AND FINITE ELEMENT ANALYSIS OF IMPACT LOADING ON COMPOSITE HONECOMB PANEL STRUCTURE

Abstract

The problem of impact loading on elastic structure due to collision with foreign object is idealized as the fundamental problem of impact loading of a relatively small sized rigid body to an elastic structure; the progression of the stresses, strains and deformation due to the loading has been modelled, developed and traced using analytical and numerical method. Based on these results, attention is then focused to the impact of a relatively rigid body to a sample beam, which are also carried out using finite element approach, which look into the static and dynamic characteristics of an impact beam attached to a structure. Comparative study has also been carried out to look into acceptable safety margin for two materials, the standard steel and Fiber Reinforced Polymer (FRP) material. With such background, analysis and finite element analysis is focussed on Fiber Reinforced Polymer (FRP) Honeycomb Sandwich Panel, noting that due to high stiffness and strength to weight ratios, composite sandwich is used increasingly in aerospace applications. A continuum damage model following work in the literature is used to model crushing due to impact. The model describes the compressive behaviour of honeycombs panel made from materials that are prone to elastic buckling. The material behaviour in compression is described by a combination of three constitutive models namely elastic, continuum damage and inelastic strain accumulation. Results from analysis are compared to existing data in the literature. The outcome in its ability to evaluate impact damage for generic honeycomb panels are assessed.