IAF MATERIALS AND STRUCTURES SYMPOSIUM (C2)
Space Structures I - Development and Verification (Space Vehicles and Components) (1)

Author: Ms. Erika Lieberknecht
United States, erika.lieberknecht@navy.mil

VERIFICATION OF A HYPERSONIC TEST PANEL THROUGH MODAL ANALYSIS OF TWO FINITE ELEMENT SOLVERS, AND VARIOUS ELEMENT TYPES AND FORMULATIONS

Abstract

Panels on hypersonic vehicles are subjected to complex aero-thermal loading, resulting in large in-plane stresses. There is a concern that these loads can lead to thermo-mechanical buckling. Since thermo-mechanical buckling can be analyzed by studying the decrease in natural frequencies of the panels as the in-plane compression and shear loads increase, investigating the panel's natural frequencies using various finite element formulations is a first step in modeling this phenomenon. Selection of appropriate shell or 3-D solid finite element, to be used for discretizing the panel is very important as the unique curvature of these panels decreases the validity of both the thin-shell as well as shallow shell assumptions. Finite element type and formulation selections for the modal analysis were investigated and comparisons were made using LS-Dyna© and Abaqus©, two well-established finite element solvers. The variation in eigenvalues and natural frequencies, obtained from various finite element formulations, of an example hypersonic panel are quantified in this study. The thermo-mechanical buckling load will next be obtained by studying natural frequencies of these panels under increased compression and shear loads.