

Modal Analysis of FGM Plates Using Finite Element Method

Authors : S. J. Shahidzadeh Tabatabaei, A. M. Fattahi

Abstract : Modal analysis of an FGM plate containing the ceramic phase of Al₂O₃ and metal phase of stainless steel 304 was performed using ABAQUS, with the assumptions that the material has an elastic mechanical behavior and its Young modulus and density are varying in thickness direction. For this purpose, a subroutine was written in FORTRAN and linked with ABAQUS. First, a simulation was performed in accordance to other researcher's model, and then after comparing the obtained results, the accuracy of the present study was verified. The obtained results for natural frequency and mode shapes indicate good performance of user-written subroutine as well as FEM model used in present study. After verification of obtained results, the effect of clamping condition and the material type (i.e. the parameter n) was investigated. In this respect, finite element analysis was carried out in fully clamped condition for different values of n . The results indicate that the natural frequency decreases with increase of n , since with increase of n , the amount of ceramic phase in FGM plate decreases, while the amount of metal phase increases, leading to decrease of the plate stiffness and hence, natural frequency, as the Young modulus of Al₂O₃ is equal to 380 GPa and the Young modulus of stainless steel 304 is equal to 207 GPa.

Keywords : FGM plates, modal analysis, natural frequency, finite element method

Conference Title : ICMMME 2016 : International Conference on Mechanical, Materials and Mechatronics Engineering

Conference Location : Madrid, Spain

Conference Dates : March 24-25, 2016