

Stress and probabilistic study of the lumbar vertebra under compression loading

Abstract

The objectives of this study were to determine the stress concentration of the lumbar vertebra and the probability of failure associated with finite element method. The lumbar vertebra model was constructed in SolidWorks software and imported by ANSYS software for the analysis. In this work, all the model components were meshed using the tetrahedral solid element (SOLID186). In order to simplify the model, all the spinal components were modeled as an isotropic and elastic material. Monte Carlo Simulation (MCS) technique was performed to conduct the probabilistic analysis using a built-in probabilistic module in ANSYS with attempt for 100 trials. The results were observed that the highest stress concentrations were found in the adjacent posterior vertebral body with 1.2117 MPa and the corresponding probability of failure for the model is 3%. Sensitivity analysis had been revealed that the force applied to the facet (FORFCT) variable was sensitive to the stress and displacement output parameters and need to be emphasized. The current probabilistic study was very useful as a tool to understand the inherent uncertainties and variations in biological structures.

Keywords — Stress analysis, finite element analysis, lumbar vertebra, Monte Carlo Simulation