Three-Dimensional Finite Element Analysis of Brain Contusion

黃豪銘

Huang;H.M.;Lee;M.C.

Abstract

A three-dimensional finite element model of the direct cortical impact experiment was built and a preliminary validation against mechanical response was completed. The motion of the impactor was enforced in the model by applying the same acceleration history as that of the experimental impactor. A nonlinear contact surface algorithm was used for impactor-brain interface with the ABAQUS general purpose finite element program. The resulting motion of the impactor and the contacting node in the brain model confirmed that the impactor moved realistically and contacted the brain surface. The pressure generated in the model compared favorably with that measured by a pressure transducer in the experiment. The pattern of high shear deformation generated at the impact site in the model was similar to the pattern of contusion hemorrhage seen in the experiment. The pressure generated at the impact site propagated to the skull-brain boundary, especially, at the posterior margin of the cerebellum. Analysis of experimental data using a biomechanically validated finite element model will enable determination of tissue-level injury criteria for application in human brain models to predict head injury potential in contact, noncontact, or side impact situations.