Finite Element Modeling of Vertebral Damage and Repair

Victor Kosmopoulos, Tony S. Keller, and Michael A.K. Liebschner*

The University of Vermont, Burlington, Vermont 05405-0156 USA;
**Rice University, Houston, Texas 77251-1892 USA;*

Vertebral damage occurs when the loads applied exceed the vertebral structural capacity (force at failure). Analytical models provide a basic understanding of the apparent forces and stresses acting on the vertebral structures, but do not tell us anything about the magnitude of tissue stresses and strains. Characterization of trabecular bone tissue stresses and strains is important in understanding basic skeletal mechanics, fracture risk, damage, and fracture repair. More explicit mechanical stress-strain behavior is obtained using numerical techniques such as the finite element analysis (FEA) method. The objective of this study was to develop a structural FEA model that could be used to simulate structural damage and repair in vertebral bodies.

A two-dimensional (2D) osteopenic FEA model was developed from sagittal μCT scan images (83 μm/element resolution) of a T10 vertebral segment. The resulting FEA model was coupled with a modulus reduction algorithm derived from perfect plasticity theory. Perfect plasticity theory uses a hyperbolic tangent function to characterize the non-linear, stress-strain behavior of vertebral trabecular bone tissue as follows: $\sigma = \sigma_0[\tanh(E_0\varepsilon/\sigma_0)^4]^{1/4}$, where σ is the tissue stress, ε is the tissue strain, σ_0 is the tissue strength (159 MPa), and E_0 (10 GPa) is the tissue modulus. The derivative of this equation with respect to strain yields an expression that can be used to characterize trabecular bone tissue damage in terms of the decrease in tissue modulus with increasing tissue strain. In the FEA models, the trabecular bone, intervertebral disc, and bone marrow tissues, are assumed to be isotropic, linear elastic materials with elastic modulus values of 10 GPa, 2.16 MPa, and 10 kPa, respectively. Previous FEA stress-strain studies have shown that model results are relatively insensitive to Poisson ratio, so a value of 0.3 is assumed for all 299 x 235 (70,265), 4-node isoparametric elements. The model is loaded using a uniform compressive load ranging from 50-500 N in 50 N increments. After each increment of load, the element stresses and strains were used to determine the modulus reduction (if any) for each bone element.

The model predicted a yield stress of 4.5 MPa and yield strain of 0.41% corresponding closely with experimental estimates of the intact strength of human thoracic lumbar vertebra. Trabecular bone tissue stress intensity (element stress/apparent stress) ranged from —5to 26. An application of this model is the simulation of vertebroplasty in which the effects of cement volume on apparent stiffness and strength, and trabecular bone tissue stress intensity are examined. Using the aforementioned model, replacing the bone marrow elements with PMMA cement elements (E = 2.16 GPa) resulted in a 2-fold increase in stiffness and a 20% reduction in the overall stress intensity. Furthermore there was over a 10-fold reduction in the number of highly stressed elements (stress intensity > 3).

In summary, the non-linear stress-strain behavior of an anatomically accurate 2D structural FEA model of an intact osteopenic vertebral body was analyzed using an iterative numerical simulation approach. The structural FEA model results reproduced experimental findings of damage in intact vertebra, and corroborates recent experimental and numerical simulations of vertebroplasty. In addition, results from this study indicate that damage to osteopenic trabecular bone occurs during body weight loading of vertebral segments.