Nanoindentation Simulation of Bovine Cortical Bone using Four- and Three- Parameter Viscoelastic/Plastic Models via Finite Element Analysis and Optimization Algorithm

Yang Zhao, Graduate Research Assistant (yzhao1@nd.edu)
Timothy Ovaert, Professor (tovaert@nd.edu)
Department of Aerospace and Mechanical Engineering, University of Notre Dame, Notre Dame, IN 46556

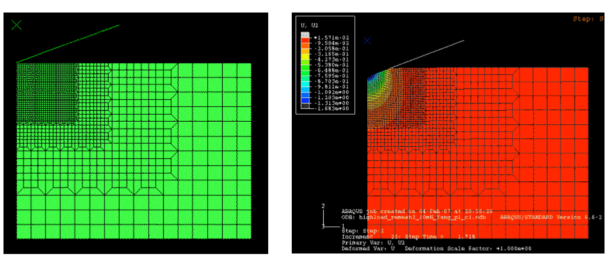
INTRODUCTION

Modeling the indentation behavior of bone is of interest for advanced characterization of its mechanical properties on both micro- and nano-scales. Studies have shown that cortical bone exhibits plastic and viscoelastic behavior in nanoindentation tests. In this study, two models (four- and three-parameter) were developed for finite element (FE) simulation of nanoindentation experimental data on bovine cortical bone. Also, an optimization algorithm was used in inverse process to facilitate automatic determination of the models' parameter.

METHODOLOGY

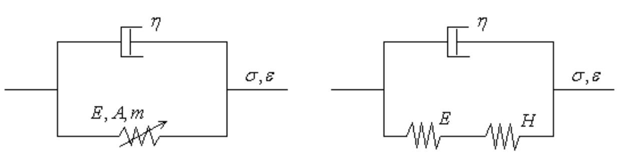
An axisymmetric FE mesh with a (rigid) Berkovich indenter (modeled as a 70.3° cone) is shown in Fig. 1(a). The mesh was refined on the corner near the cone using the commercial software Patran™.

Four- and three-parameter viscoelastic/plastic constitutive models (Fig. 2) were implemented as user subroutines in ABAQUS™ to define the material properties of the elements. The four-parameter model is composed of an elasto-plastic spring, with parameters *E* (elastic stiffness), and *A*, *m* (power law plasticity), in parallel with a linear dashpot or viscosity η. The three-parameter model simplifies the plastic deformation using a linear hardening modulus, *H*.



(a) (b)

Fig. 1. (a) FE mesh, and (b) simulation results (b)



(a) (b)

Fig. 2. (a) Four-parameter, and (b) three-parameter constitutive models

The values of the parameters were determined using an inverse approach implemented via a Matlab™ optimization program. First, Matlab™ uses an initial guess of the parameter values to run ABAQUS™ (using a parametric study method in ABAQUS™), generating a simulated indentation load vs. displacement curve (Fig. 4). Then, an optimization process ensues, as shown in Fig. 3.

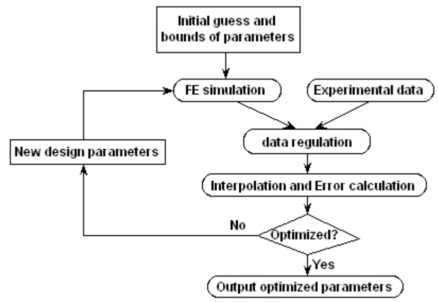
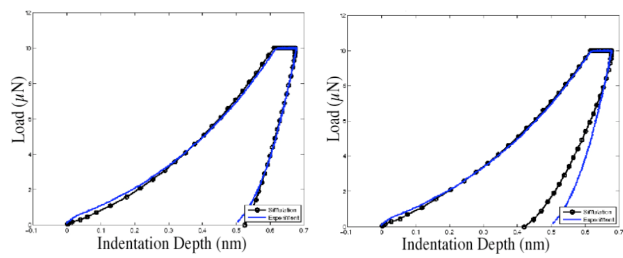


Fig. 3. Optimization algorithm flow-chart

The FE and experimental curves are regulated and compared, and the error is defined. Finally, the values of the parameters are optimized through an iteration loop.

RESULTS

Fig. 1(b) shows the displacement field at maximum load of 10mN. The results are typical for conical indentation [1]. The parameters are: E = 19.8 GPa, A = 549, m = 2.4, η = 2.4 GPa•s, for the four-parameter model (Fig. 4(a)). The three parameter model yields E = 17.0 GPa, H = 9.0 GPa, and η = 3.5 GPa•s (Fig. 4(b)). These values (particularly *E*) are reasonable based on published bone data [2]. For the three-parameter model, we focused on the initial unloading curve, thus the final deformation is prioritized in the optimization.



(a) (b)

Fig. 4. Optimization results for (a) four-parameter model, and (b) three-parameter model. Blue: experiment, black: simulation.

CONCLUSION

The agreement between the simulation and experimental curves demonstrates the validity of the optimization technique and the parametric material models. The goal is to provide better insight into the indentation behavior of bone using computational methods, and to use the results to develop new hierarchical models at multiple length scales which capture the heterogeneous nature of bone itself.

REFERENCES

[1] Johnson, K.L., 1995, Contact Mechanics, Cambridge.
[2] Zhang, J., et al., 2008, *J. Biomech.*, **41**, pp. 267-275.