BONE FRACTURE ANALYSIS USING THE EXTENDED FINITE ELEMENT METHOD (XFEM) WITH ABAQUS

Xiangyi (Cheryl) Liu, Xiaoliang Qin and Zhen-zhong Du

Dassault Systemes Simulia Corp, Providence, RI, USA
email: Cheryl.LIU@3ds.com web: www.simulia.com

INTRODUCTION

The bones of elderly people with osteoporosis are susceptible to either traumatic fracture as a result of external impact, such as what happens during a fall, or even spontaneous fracture without trauma as a result of muscle contraction [1, 2]. Understanding the fracture behavior of bone tissue will help researchers find proper treatments to strengthen the bone in order to prevent such fractures, and design better implants to reduce the chance of secondary fracture after receiving the implant.

A number of fracture criteria have been proposed for bone tissue and many of the studies used FEA models to correlate critical values of the proposed criteria with bone fracture patterns observed in experiments [3, 4]. However, simulation of actual crack initiation and growth has been hard to achieve using the conventional FEA approach. With the new extended finite element method (XFEM) available since Abaqus 6.9, researchers can simulate crack initiation and growth more easily. In this study we demonstrate how XFEM can be used to predict proximal femur fracture due to impact.

METHODS

Modeling stationary discontinuities, such as a crack, with the conventional finite element method requires that the mesh conform to geometric discontinuities. Therefore, considerable mesh refinement is needed in the neighborhood of the crack tip to capture the singular asymptotic fields adequately. Modeling a growing crack is even more cumbersome because the mesh must be updated continuously to match the geometry of the discontinuity as the crack progresses.

The extended finite element method was first introduced by Belytschko and Black [5]. It is an extension of the conventional finite element method based on the concept of the partition of unity by Melenk and Babuska [6], which allows local enrichment functions to be easily incorporated into a finite element approximation. The presence of discontinuities is ensured by the special enriched functions in conjunction with additional degrees of freedom. However, the finite element framework and its properties such as sparsity and symmetry are retained [7].

The geometry of an intact human femur was imported into Abaqus/CAE and meshed with linear tetrahedron elements (C3D4). The bone density distribution (Fig. 1) was obtained using a bone remodeling algorithm implemented with Abaqus subroutine USDFLD [8].

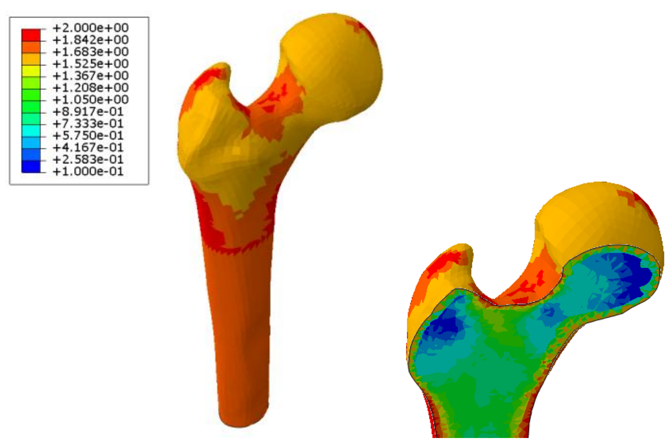

Figure 1: Bone density distribution predicted by bone remodeling theory.

The bone density is incorporated into the model as a field variable. The bone material is assumed to be isotropic linear elastic with Young's modulus as a cubic function of bone density. Similarly, the damage initiation and damage evolution parameters are also defined as functions of bone density. Maximum principal strain is used as the damage

initiation criterion. Damage evolution is assumed to follow the energy dissipated during the process.

A femur fracture test is simulated using the Abaqus implicit dynamic procedure with the top plate fixed, the distal end of the femur and the bottom supporting plate moving up toward the top plate at the same speed (Fig. 2).

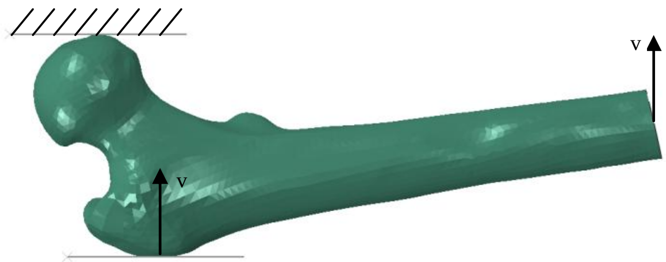

Figure 2: Boundary conditions of the FEA model.

RESULTS AND DISCUSSION

The Abaqus XFEM analysis predicted crack initiation in the femur neck just below the femur head (Fig. 3a). The crack grew toward the femur head (Fig. 3b, 3c) and eventually crossed from the femur neck into the femur head (Fig. 3d).

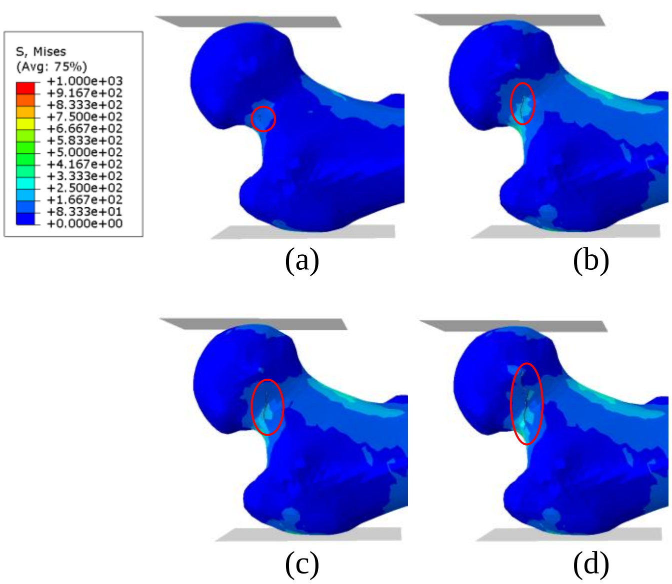

Figure 3: Analysis results.

Since no physical testing was performed for this specific femur, we were not able to validate the analysis results. However, similar bone fracture patterns were observed in experiments [9]. To properly validate such an analysis, the initial bone density should be calculated based on CT data.

CONCLUSIONS

Bone, as a biological tissue, possesses a very complex hierarchical structure. From a mechanical point of view, it is transversely isotropic with a higher modulus in the longitudinal direction. It is asymmetric with higher strength in compression than in tension and shear. The fracture and failure properties of bone tissue are even more complex. Various theories have been proposed including stress or strain-based criteria, von Mises or maximum principal invariant-based criteria, and more recently composite failure criteria.

What we implemented in this study is a very simplified version of bone material properties. Our goal was to demonstrate the new XFEM technology and how it may be used to study bone fracture and failure properties. In Abaqus 6.10-EF1, a new user subroutine permitting user-defined damage initiation criteria will be available for Abaqus XFEM, which will provide greater flexibility in implementing new failure criteria. In future studies, we will demonstrate how composite failure criteria may be implemented using this user subroutine to study bone fracture.

REFERENCES

1. Sloan J, et al. *Injury* **13**, 230-2, 1981.
2. Horiuchi T, et al. *Orthopedics* **11(9)**, 1277-80, 1998.
3. Keyak JH, et al. *J Biomech*, 31, 125-33, 1998.
4. Oden ZM, et al. *J Orthop Res*, 17(5), 661-7, 1999.
5. Belytschko, et al. *Int J Numer Meth Eng*, 45, 601-20, 1999.
6. Melenk J, et al. *Comput Method Appl M*, 39, 289-314, 1996.
7. Abaqus 6.9 Documentation, Analysis User's Manual, Section 10.6.1.
8. Liu X, et al, *ASME Summer Bioengineering Conference*, Naples, FL, 2010 (under review).
9. Shaw MN, et al. *Mater Sci Tech*, 3, 1457-1467, 2007.