Abaqus Technology Brief

TB-10-BONE-1

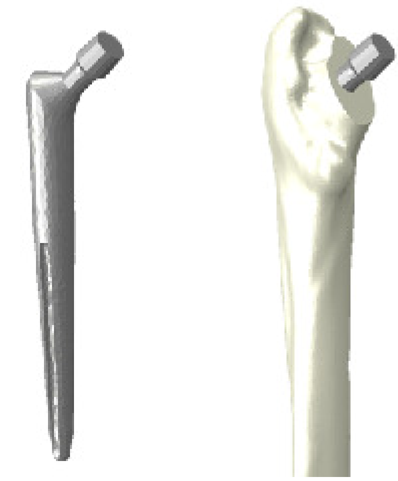
Revised: March 2010

Simulation of Adaptive Bone Remodeling with Abaqus/Standard

Summary

In the adaptive bone remodeling process, the density of bone tissue changes over time according to the load it sustains. Elevated loads produce increases in bone density while reduced loads cause reduction of bone density. The long term success of an orthopedic implant can be better predicted by including this process in the design workflow.

In this Technology Brief, we demonstrate the Abaqus/Standard implementation of one of the leading bone remodeling algorithms. User subroutine USDFLD is employed to capture solution dependent material properties, and the approach is used in the analysis of a total hip replacement design.



Key Abaqus Features and Benefits

- Spatial and temporal variation of material properties through the use of field variables
- User subroutine USDFLD to allow solution-dependent material properties
- Robust pre-processing capability of Abaqus/CAE to import and modify existing CAD geometries

Background

Healthy bones adapt to mechanical stimulus in a process known as bone remodeling. More specifically, the apparent density of bone tissue is directly related to the load it bears over time. For example, after long periods in a microgravity environment astronauts experience a loss of bone density because of skeletal unloading, while most professional tennis players experience a gain of bone density in their playing arm because of the repetitive loading applied over many years.

The bone remodeling process is an important concern in the orthopedic implant industry. When an implant is used to treat trauma or chronic osteoarthritis, a portion of the bone is replaced. The altered stress state in the bone leads to a remodeling of the remaining bone tissue. Since implants are made of metals such as titanium and cobalt-chromium, they are much stiffer than the surrounding bone tissue; they subsequently bear more load than the bone they replace. As a result, the remaining bone tissue can eventually lose most of its structure. This effect, known as stress shielding, can cause loosening of the implant and eventually failure of the fixation. Hence, the extent to which the bone tissue is able to maintain its apparent density is an important measure of long-term implant success.

The slow pace of bone remodeling adds to the difficulty of experimental studies; for this reason and others, simulation through finite element analysis is an ideal way to study the process. For a numerical analysis to be realistic, it must incorporate a proper remodeling theory. Among the many available, the strain adaptive remodeling approach developed by Huiskes, *et al*. [1] has been compared with *in vivo* studies with moderate success and has been used by many researchers to compare implant designs.

In this Technology Brief, the theory presented in [1] is implemented in Abaqus and a remodeling simulation is completed. Only a single analysis is required, without the need for complicated post processing efforts. This approach facilitates the inclusion of bone remodeling simulation in the workflows of implant designers, researchers, and physicians.

Analysis Approach

Implementation of the bone remodeling algorithm

Bone tissue exhibits a direct relationship between its density and mechanical properties. Density can be calculated with data obtained from medical images such as CT scans. The mechanical properties of the tissue comprising a complete bone will vary not only spatially, but also temporally as the tissue density changes with the remodeling process. In this study a cubic relationship between the bone density and Young's modulus was defined

based on the work of Carter and Hayes [2]. The spatial non-uniformity was captured in Abaqus by making the material properties dependent on field variables.

In the strain adaptive bone remodeling theory, bone tissue density is a function of strain energy density. Abaqus user subroutine USDFLD, which allows for the definition of solution-dependent field variables at an element integration point, is ideal for such situations.

In this approach, material properties are defined with field variable dependence; the field variable is computed in the subroutine based on the solution from the previous increment. Thus by treating the bone density as the field variable to be computed in USDFLD, the bone mechanical properties are allowed to vary with density.

A schematic relationship between bone density and remodeling signal is shown in Figure 1. In particular, a "lazy zone" exists around a reference signal level; this represents an inactive region in which bone density does not change. When the remodeling signal is above or below the lazy zone, bone density starts to increase or decrease accordingly. In this study, strain energy density per unit mass was used as the remodeling signal. The rate of density change depends on user-chosen constants and the representative area of the bone tissue, which is a function of bone density.

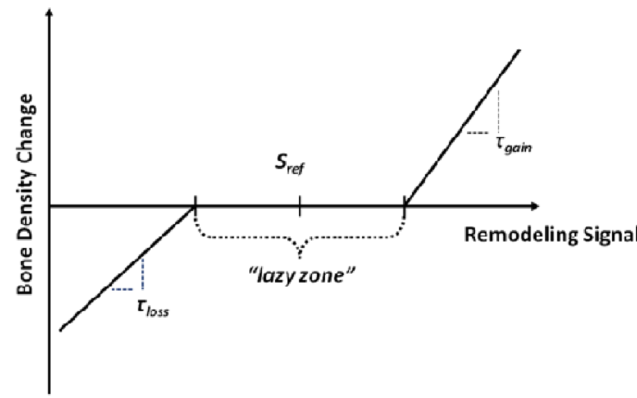


Figure 1: Bone density change as a function of the modeling signal

Total Hip Replacement Model

In this application, the geometry of an intact human femur bone and a total hip replacement stem were imported into Abaqus/CAE. An extrusion cut was made to the femur to remove the femur head. This operation mimics the surgical procedure of a total hip replacement. Both the femur and the implant were meshed with second order tetrahedral elements (C3D10M).

Realistic boundary conditions were applied to the proximal and distal ends of the femur-implant assembly [3]. Abductor and joint forces were applied to the control points of coupling constraints, which distribute the forces to affected regions. Rough contact was defined between the implant and bone tissues, assuming the existence of the full bonding that results from bone ingrowth into the porous scaffold on the implant surface. Initial bone density distribution was applied as an initial condition. The exterior of the femur head and femur shaft was assumed to be cortical bone, while the inside of the femur head was assumed to be trabecular bone. Two loading conditions and two implant materials were analyzed.

Results and Discussion

Predicted bone loss patterns with different implant materials and loading magnitudes are shown in Figure 2. Elements in which bone density is lost have been removed from the plots. Clinical observations show that most bone loss due to stress shielding happens near the lesser trochanter area [4]; this is verified by the simulation results, which show bone loss in the interior region around the lesser trochanter (Figure 2a).

Stress shielding is known to be caused by the stiffness difference between the bone and implant. The simulation results show that when the implant material is changed from titanium, which is much stiffer than bone tissue, to a softer polymer material, there will be significantly less bone loss (Figure 2b). Combined with a fatigue analysis, an implant designer will be able to use the current workflow to determine an optimized implant modulus that provides less bone loss and longer implant fatigue life.

Bone loss is also affected by the magnitude of the applied loading. Figure 2c shows the titanium implant result with a 35% load reduction. The simulation results showed more bone loss than with the full load (Figure 2a), especially in the exterior region of the femur. This is why patients are encouraged to move as much as they can after surgery so that bone density is maintained. It also suggests that in situations where the patient is restrained from vigorous exercise, a different implant design may result in less bone loss.

Conclusions

Abaqus/Standard provides the tools necessary to model solution dependent material properties. As applied to orthopedic implants, this capability allows for the inclusion of adaptive bone remodeling in the design process. Physicians, researchers, and designers are thus afforded a tool to compare the performance of different implant designs under different loading conditions via virtual experiments.

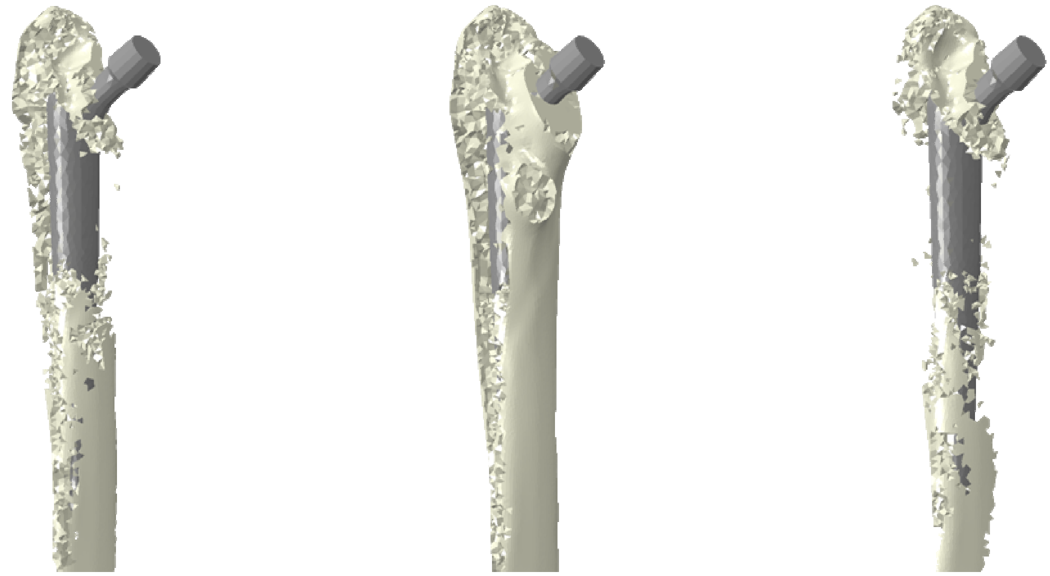


Figure 2: Predicted bone loss patterns - (a) an implant made of titanium, (b) an implant made of ultra high molecular weight PMMA, and (c) the titanium implant with 35% load reduction

Acknowledgements

The authors would like to thank Dr. Jeff E. Bischoff of Zimmer Inc. for providing the implant geometry.

References

1. R. Huiskes, H. Weinans, H. J. Grootenboer, M. Dalstra, B. Fudala and T. J. Slooff, "Adaptive bone-remodeling theory applied to prosthetic-design analysis," Journal of Biomechanics, 20(11/12): 1135-50, 1987
2. D. R. Carter and W. C. Hayes, "The compressive behavior of bone as a two-phase porous structure," Vol. 59, pp. 954-962, 1977
3. A. D. Speirs, M. O. Heller, G. N. Duda and W. R. Taylor, "Physiologically based boundary conditions in finite element modeling," Journal of Biomechanics, Vol.40, pp. 2318-2323, 2007
4. W. L. Turner, R. M. Gillies, R. Sekel, P. Morris, W. Bruce and W. R. Walsh, "Computational bone remodeling simulations and comparisons with DEXA results," Journal of Orthopaedic Research 23 (2005) 705-712

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, please see the following Abaqus 6.13 documentation references:

- Analysis User Subroutines Reference Guide
  - USDFLD: User subroutine to redefine field variables at a material point, Section 1.1.50