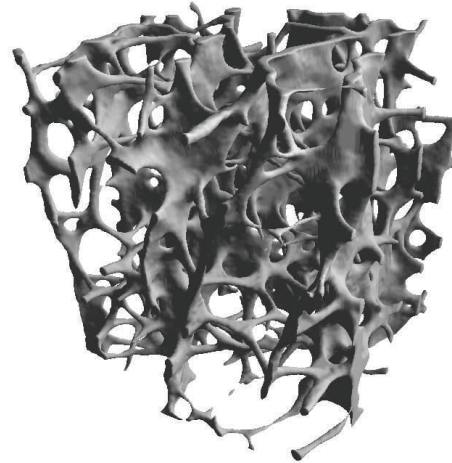


Nonlinear Micro Finite Element Analysis of Human Trabecular Bone

Summary

Trabecular bone must withstand the loads that arise during daily activities as well as those due to trauma. Investigation of the mechanical properties of trabecular bone presents a challenge due to its high porosity and complex architecture, both of which vary substantially between anatomic sites and across individuals. While Micro Finite Element (μ FE) analysis of trabecular bone is the most commonly used method to analyze trabecular bone mechanical behavior, the large size of these models has forced researchers to use custom codes and linear analysis. The nonlinear capabilities in Abaqus allow efficient analysis of these models and provide answers to important research questions.



Background

Trabecular bone found at the end of long bones (e.g., femur) and in cuboidal bones (e.g., spine) is a major load bearing biological tissue in the human skeleton. Its mechanical properties are of great clinical and research interest. Improved understanding of trabecular bone mechanical properties will provide insight into fracture mechanisms in bones as well as allow the assessment of the effects of aging, disease, and drug treatments. Trabecular bone is a highly porous tissue—with over 85% porosity in the spine—with a complex architecture that varies substantially between anatomic sites and across individuals (Figure 1). Therefore, mechanical property data from multiple specimens are required to determine statistically the mechanical properties of trabecular bone.

Micro finite element models (μ FE) are used extensively to study the mechanical properties of trabecular bone, both at the continuum level and at the microstructural level. These models are obtained through high-resolution imaging of trabecular bone specimens that can be automatically converted into finite element meshes with hexahedral elements (Figure 2). All elements in these meshes are identical and are typically 50 μ m in size. At this discretization level, a 5 mm cubic specimen μ FE model will typically have over half a million degrees of freedom. μ FE models of bone specimens that are similar to experimentally used specimens (8 mm diameter and 15 mm length) contain millions of degrees of freedom.

In the past the large scale of these problems forced many researchers to use custom codes that utilize element-by-element iterative solvers. Due to the complexity of nonlinear finite element modeling, these custom codes have been limited to linear elastic analysis. Although linear

Key Abaqus Features and Benefits

- Constitutive models for simulating the bone tissue mechanical behavior
- The ability to solve both geometrically and materially nonlinear models easily
- Parallel solver capability

elastic finite element models cannot simulate failure behavior of bone, they are widely used by researchers to determine bone tissue elastic properties by calibrating them against experimental data (Ref. 1). However, many questions remain unanswered regarding nonlinear mechanical behavior of trabecular bone.

Abaqus/Standard is well suited to these types of analyses because it can solve large problems using parallel execution and includes sophisticated material models. In this technology brief we investigate the role of geometrical nonlinearities in trabecular bone mechanical behavior using Abaqus/Standard. We also demonstrate the feasibility of solving large problems by examining the parallel performance (i.e., scalability) of a single linear elastic analysis for a model containing over 4 million degrees of freedom.

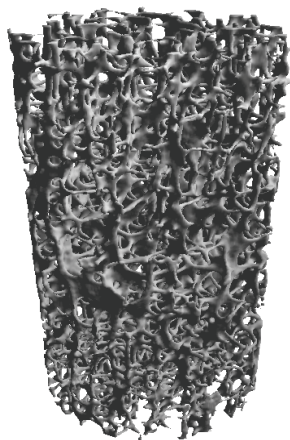


Figure 1: Rendering of a cylindrical trabecular bone specimen from the human spine ($\varnothing=8$ mm, $L=15$ mm length)

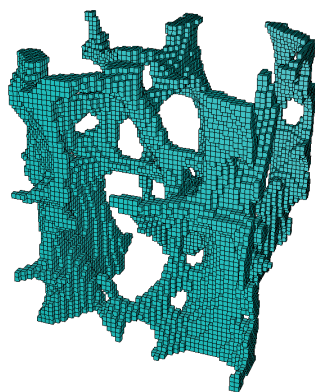


Figure 2: μ FE mesh detail of 2.5 mm cubic portion of the bone specimen with 44 μ m elements.

The cylinder model was used to assess the parallel performance of the direct sparse solver. Frictionless, displacement boundary conditions were applied at the top and bottom surfaces to simulate a compressive 1% strain. Linear elastic analyses were conducted using 1, 2, and 4 CPUs of an HP rx8620 computer.

For nonlinear analysis the 5 mm cube model was used. A cube of this dimension is large enough to determine continuum level properties but small enough to make nonlinear analysis feasible. Bone tissue was modeled using the cast iron plasticity material model. Cast iron plasticity provides elastic-plastic behavior with different yield strengths and hardening in tension and compression and results in an unsymmetrical element stiffness matrix. Therefore, the parallel sparse direct solver with unsymmetric storage was used. A tissue elastic modulus of 13.4 GPa was used with a Poisson's ratio of 0.3 (Ref. 2). For the cast iron plasticity model, tissue yield stresses were 55.2 MPa in tension and 110.6 MPa in compression, based on the yield strains reported for human femoral trabecular tissue (Ref. 3). In both tension and compression, a hardening slope equal to 5% of the elastic modulus was used. Frictionless displacement boundary conditions were used to apply a 2% nominal strain in tension and compression. At such a low level of nominal strain, self-contact of the bone microstructure need not be considered. In addition, each model was run with and without the effects of geometrically nonlinear deformations. In total, four nonlinear analyses were performed and continuum level yield strains were calculated for comparison. All the analyses of the cube were performed using 2 CPUs of an IBM Power4 computer.

Abaqus/Standard is well suited to these types of analyses because it can solve large problems using parallel execution and includes sophisticated material models. In this technology brief we investigate the role of geometrical nonlinearities in trabecular bone mechanical behavior using Abaqus/Standard. We also demonstrate the feasibility of solving large problems by examining the parallel performance (i.e., scalability) of a single linear elastic analysis for a model containing over 4 million degrees of freedom.

Finite Element Analysis Methodology

One human vertebral trabecular bone specimen with 9% volume fraction was imaged using micro-computed tomography (μ CT 20, Scanco Medical AG, Bassersdorf, Switzerland) at 22 μ m resolution (Figure 1). Two μ FE models were created. First, the whole cylindrical specimen was meshed with hexahedral elements of size 44 μ m (Figure 2). Second, a 5 mm cubic subregion from the core of the cylinder was used to create a model with the same element size. Mesh quantities for both models are listed in Table 1.

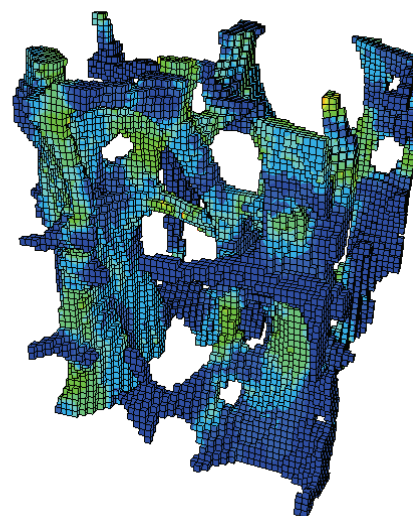
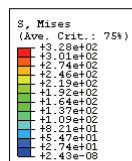


Figure 3: Mises stress in 2.5 mm cubic portion of μ FE model subjected to 2% compressive strain showing the localized stress distribution in the bone structure.

Results and Conclusions

Linear analysis of the cylinder model took under 16 minutes wall clock time on 4 CPUs and used less than 11 GB of memory (Table 2). The parallel direct solver scaling results are also shown in Table 2; the speed-up factor is based on the solve time. Nonlinear analyses of the cube μ FE model with geometrical nonlinearities took less than 7.4 hours wall clock time and required 4.1 GB memory. Each nonlinear analyses required approximately 100 linear equation solves emphasizing the importance of the solver scalability. The localization of initial yielding within the bone structure makes convergence of nonlinear analyses particularly challenging (Figure 3).

The apparent stress (applied force/cross-sectional area = 25 mm^2) is plotted against apparent strain (change in specimen length/original specimen length) in Figure 4.

Initial yield is defined as the point at which 0.2% offset is reached. Similar to reported experimental data (Ref. 4), yield strains were higher in compression than tension.

Although the tissue material was hardening, softening was observed at the apparent level when geometric nonlinearities were included (Figure 4). In addition, the yield strains were similar to experimentally measured values, particularly in compression (Ref. 4). These results show that the different yield behaviors of the trabecular tissue in tension and compression and geometric nonlinearities need to be incorporated in μ FE models to model the continuum level yield behavior of trabecular bone accurately.

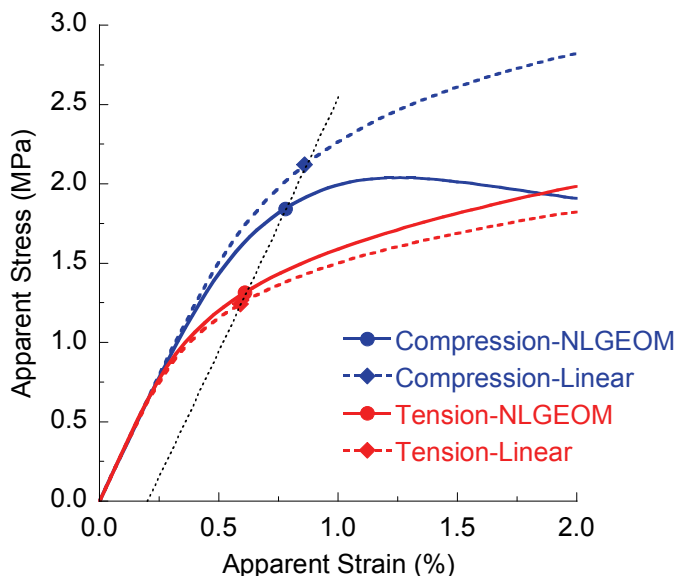


Figure 4: Stress-strain behavior for the four nonlinear analyses. The effects of geometric nonlinearities cause softening in compression and stiffening in tension. Markers show initial yield points determined using the 0.2 offset method (dotted line).

Table 1: Mesh quantities for μ FE models

Model	Number of Elements	Number of Nodes	Number of Degrees of Freedom
Cylinder	828,853	1,380,834	4,142,502
Cube	131,322	216,027	648,081

Table 2: Version 6.4-3 direct sparse solver performance

Number of CPUs	Solve Time (s)	Speed-Up	Total Wall Clock Time (s)
1	554	1.00	1,348
2	295	1.88	1,107
4	171	3.24	945

Table 3: Yield strains for different modeling combinations

Geometrical Nonlinearity	Loading Mode	
	Tension	Compression
ON	0.61	0.78
OFF	0.59	0.86

Acknowledgements

Dassault Systèmes SIMULIA Corp. would like to acknowledge Professor Tony M. Keaveny of the University of California, Berkeley, for providing the bone specimen image data and finite element mesh.

References

1. van Rietbergen, B.; H. Weinans; R. Huiskes; A. Odgaard, "A New Method to Determine the Trabecular Bone Elastic Properties and Loading Using Micromechanical Finite Element Models," *Journal of Biomechanics*, vol. 28, pp. 69–81, 1995.
2. Rho, J. Y.; T. Y. Tsui; G. M. Pharr, "Elastic Properties of Human Cortical and Trabecular Lamellar Bone Measured by Nanoindentation," *Biomaterials*, vol. 18, pp. 1325–1330, 1997.
3. Bayraktar, H. H.; E. F. Morgan; G. L. Niebur; G. E. Morris; E. K. Wong; T. M. Keaveny, "Comparison of the Elastic and Yield Properties of Human Femoral Trabecular and Cortical Bone Tissue," *Journal of Biomechanics*, vol. 37, pp. 27–35, 2004.
4. Morgan, E. F.; and T. M. Keaveny, "Dependence of Yield Strain of Human Trabecular Bone on Anatomic Site," *Journal of Biomechanics*, vol. 34, pp. 569–577, 2001.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief, see the following Abaqus 6.10 documentation references:

- Analysis User's Manual
 - "Static stress analysis," Section 6.2.2
 - "Parallel execution in Abaqus/Standard," Section 3.5.2
 - "Cast iron plasticity," Section 20.2.10

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Suresnes, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include ABAQUS, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright © 2007 Dassault Systèmes

