SCI INSTITUTE

TECHNICAL REPORT

A Comparison of FEBio, ABAQUS, and NIKE3D Results for a Suite of Verification Problems

Steve A. Maas, Benjamin J. Ellis, David S. Rawlins and Jeffrey A. Weiss

UUSCI-2009-009

Scientific Computing and Imaging Institute
University of Utah
Salt Lake City, UT 84112 USA

December 1, 2009

Abstract:

Computational modeling has become a standard methodology in biomechanics, both for interpreting experimental results and as an investigative approach. The finite element (FE) method is by far the most common numerical technique that is used for this purpose. Investigators have primarily used commercial software, such as ABAQUS, that is neither geared toward biological applications nor sufficiently flexible to follow the latest developments in the field. This lack of a tailored software environment has hampered research progress, as well as dissemination of models and results. To address these issues, we developed FEBio (http://mrl.sci.utah.edu/software.php?menu=Software), a nonlinear implicit FE framework, designed specifically for analysis in computational solid biomechanics. Further, we have created a public forum (http://mrl.sci.utah.edu/forums) were models and results are disseminated. The current test suite for FEBio consists of over 100 test problems. The results from FEBio, ABAQUS, and NIKE3D for representative problems from the test suite are presented here, showing the nearly identical solutions produced by the three codes. This excellent agreement should give researchers confidence going into a new research problem that FEBio can provide accurate results. Finally, we continue to develop FEBio. New element formulations and constitutive equations are being implemented and will become available in future versions of the code.

THE UNIVERSITY OF UTAH

Introduction

The finite element (FE) method is by far the most common numerical discretization and solution technique that has been used in computational biosolid mechanics. The FE method provides a systematic method for assembling the response of a complex system from individual contributions of elements, and thus is ideal for the complex geometries often encountered in biomechanical systems. It also provides a consistent way to address material inhomogeneities and differences in constitutive models between disjoint or continuous parts of a model. The solution procedure involves the consideration of overall energy minimization and/or other fundamental physical balance laws to determine unknown field variables over the domain. The FE method has been applied to problems in biomechanics as early as the 1970s (e.g. [1-6]). The application of finite element analysis in biomechanics research and design has increased exponentially over the last 30 years as commercial software availability has improved and researchers obtained better access to appropriate computing platforms. Applications have spanned from the molecular to cellular, tissue and organ levels.

However, the lack of a FE software environment that is tailored to the needs of the field has hampered research progress, dissemination of research and sharing of models and results. Investigators have primarily used commercial software, but these packages are not geared toward biological applications, are difficult to verify, preclude the easy addition and sharing of new features such as constitutive models, and are not sufficiently general to encompass the broad framework needed in biomechanics. To address these issues, we developed FEBio (an acronym for "Finite Elements for Biomechanics"), a nonlinear implicit finite element framework designed specifically for analysis in computational solid biomechanics [7].

Arguably the most important aspect of developing a new FE code is proper verification. The American Society of Mechanical Engineer's 'Guide for Verification and Validation in Computational Solid Mechanics' [8] defines verification as:

> 'The process of determining that a computational model accurately represents the underlying mathematical model and its solution. In essence, verification is the process of gathering evidence to establish that the computational implementation of the mathematical model and its associated solution are correct.'

In the case of computational solid biomechanics, the mathematical model is based on the partial differential equations that arise from the equations of motion (conservation of linear momentum), the associated boundary conditions, initial conditions, and constitutive equations. Development of a numerical method of analysis based on the mathematical model requires numerical discretization, solution algorithms, and convergence criteria [8, 9]. To verify the numerical methods and computational implementation of the mathematical model in FEBio, it must be demonstrated that it gives the correct solution to a set of benchmark problems that consist of either analytical solutions or results from established FE codes, such as ABAQUS and NIKE3D. We chose to compare the solutions from FEBio to those of ABAQUS and NIKE3D because ABAQUS is the most utilized FE code in the field of biomechanics and we have over 15 years experience using NIKE3D.

Our long term goal is to develop a freely available, extensible finite element modeling framework for solid mechanics, fluid mechanics, solute transport, and electrokinetics in biological cells, tissues and organs, based around the FEBio framework. To date, no such tools are available for general use in the public domain. The specific objectives of this report are to present results for a suite of verification problems to verify FEBio and simultaneously illustrate some of its capabilities.

Verification Problems

Results produced by FEBio for a suite of verification problems were compared to the solutions from ABAQUS (Version 6.7, Simulia, Providence, RI) and NIKE3D (Version 3.4.2). The methods (problem description) and results for each verification problem are presented together in the following sections. The current test suite for FEBio consists of over 100 test problems, and the problems described below are representative.

Strip Biaxial Stretching of an Elastic Sheet with a Circular Hole. A thin, initially square sheet containing a centrally located circular hole was subjected to uniaxial stretching. The example demonstrates the use and verifies the results of a hyperelastic material in plane stress. The undeformed square sheet was 2 mm thick and 165 mm on each side with a centrally located internal hole of radius 6.35 mm (Fig. 1). The sheet was meshed with 32 three-field elements.

Plane stress boundary conditions were defined for the in-plane faces of the sheet. The sheet was stretched to a width of 1181 mm using nodal displacements, while the edges parallel to the displacement were restrained from contracting using nodal constraints. The sheet material was represented by a

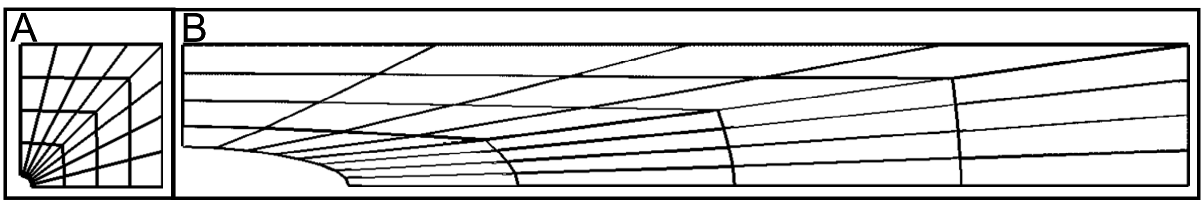


Figure 1. A) Underformed mesh. B) Deformed mesh.

hyperelastic Mooney-Rivlin constitutive model with uncoupled deviatoric and volumetric behaviour. The uncoupled strain energy W is given by:

$$W = C_1\left(\tilde{I}_1 - 3\right) + C_2\left(\tilde{I}_2 - 3\right) + \frac{1}{2}K\left(\ln J\right)^2 \qquad (1)$$

C_1 and C_2 are the Mooney-Rivlin material coefficients, $\tilde{I}_1$ and $\tilde{I}_2$ are the invariants of the deviatoric part of the right Cauchy-Green deformation tensor, $\tilde{\boldsymbol{C}} = \tilde{\boldsymbol{F}}^T\tilde{\boldsymbol{F}}$, where $\tilde{\boldsymbol{F}} = J^{(-1/3)}\boldsymbol{F}$, **F** is the deformation gradient, K is the bulk modulus, and $J = \det\left(\boldsymbol{F}\right)$ is the Jacobian of the deformation. When $C_2 = 0$, this model reduces to an uncoupled version of the incompressible neo-Hookean constitutive model in Eq. (1). For this problem, the Mooney-Rivlin coefficients and bulk modulus were $C_1 = 0.1863$ MPa, $C_2 = 0.00979$ MPa, and K = 100 as first used by Oden [10] to match the experimental results for a rubber sheet first reported by Treloar [11]. The reaction force vs. percent elongation predicted by FEBio was compared to results from ABAQUS and NIKE3D.

The total nodal reaction force predicted by FEBio was identical to the results produced by NIKE3D, but varied some from ABAQUS due to the different algorithms used to enforce the material incompressibility (Fig. 2). FEBio and NIKE3D use an augmented-Lagrangian routine to enforce isochoric deformation while ABAQUS uses a Lagrangian multiplier routine.

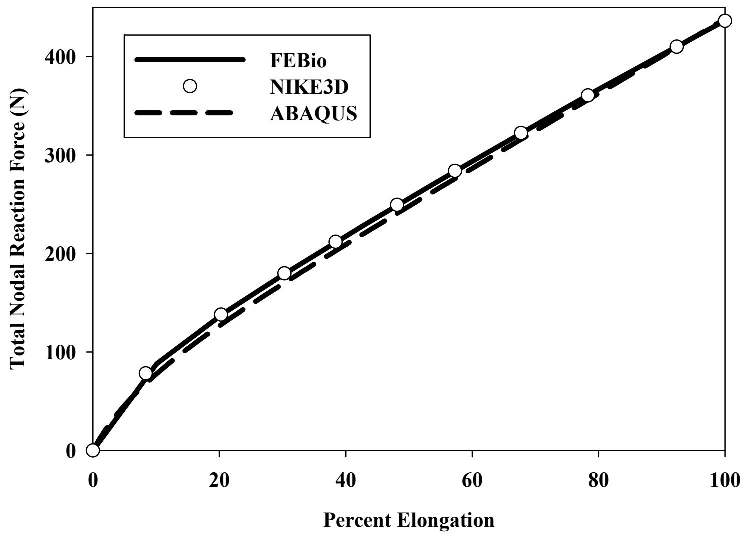


Figure 2. Total nodal reaction force versus percent elongation results for FEBio, ABAQS, and NIKE3D

Upsetting of an Elastic Billet. The compression of a billet of material with two rigid flat surfaces was analyzed using only nodal constraints and displacements as the boundary and loading conditions. This is a simplification of the contact problem presented next because the material is only compressed to the point when the material bulging from the middle of the billet makes contact with the flat surfaces (Fig. 3, B). The results from this problem were used to further verify the results from the contact problem presented next (Fig. 3, C).

A quarter-symmetry, plane strain model of a cube of material was produced for this problem (Fig. 3, A). The model dimensions were 1 X 1 X 0.1 mm and a butterfly mesh with 10 elements per outer side and one element through the thickness produced converged results. Boundary and loading conditions were prescribed using nodal constraint and nodal loads, respectively. The billet was represented as a Mooney-Rivlin material with C1 = 1 MPa, C2 = 10 MPa, and bulk modulus (K) = 10,000 MPa. An augmented-Lagrangian routine was used to enforce an isochoric deformation.

The displacement of the top right node of the mesh (the billet bulge) was plotted versus the

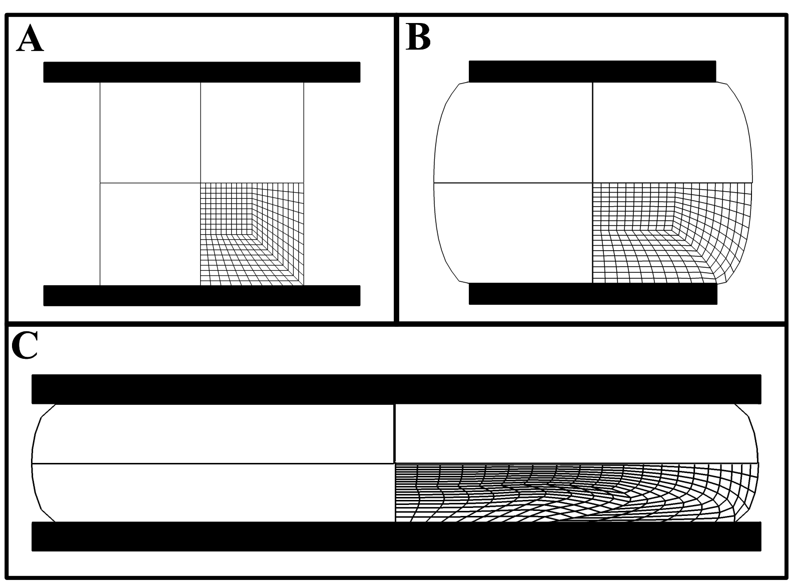


Figure 3. A) Undeformed billet mesh. B) Deformed non-contact billet. C) Deformed billet with contact.

percent compression and the results from FEBio were compared to results from ABAQUS and NIKE3D.

The lateral displacement of the billet predicted by FEBio was nearly identical to the results produced by ABAQUS and NIKE3D (Fig. 4).

Billet with Contact. This problem adds the complexity of contact between an incompressible material and a rigid plane. The billet was compressed 60% of its original height causing extensive contact between the material originally in the middle of the billet and the rigid plane. The model geometry, material, and boundary conditions for this problem were the same as the previous problem, but for this problem the compression of the billet was caused by frictionless contact between the billet and rigid plane. This contact was enforced using an augmented-Lagrangian routine.

The displacement of the top right node of the mesh (the billet bulge) was plotted versus the percent compression and the results from FEBio were compared to results from ABAQUS and NIKE3D.

The lateral displacement of the billet predicted by FEBio was nearly identical to the results produced by ABAQUS and NIKE3D and furthermore the results for both types of problem line up through the smaller range of compression (Fig. 4).

Crushing of a Pipe. This problem simulated the crusing of a long, straight pipe of cylindrical cross-section between two flat, frictionless anvils. This benchmarking problem includes large rotations and contact between deformable and rigid materials. A quarter-symmetry, plane strain model of a 114.3 mm radius pipe with an 8.87 mm thickness was created for this problem (Fig. 5). The model was meshed

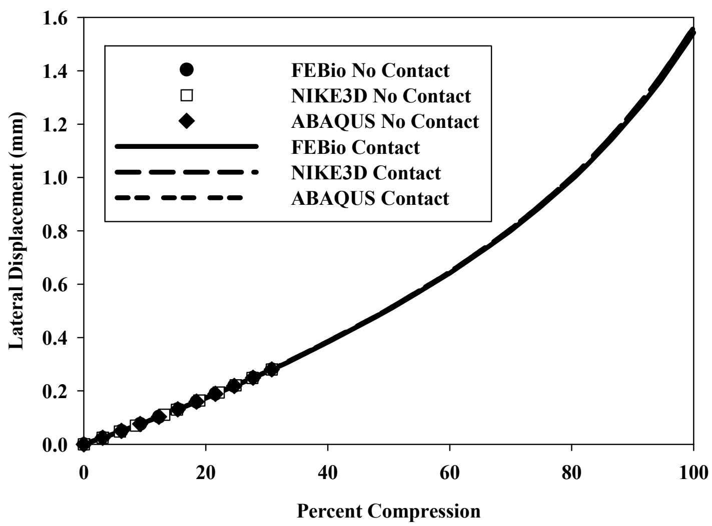


Figure 4. Contact and non-contact billet lateral bulge versus percent compression.

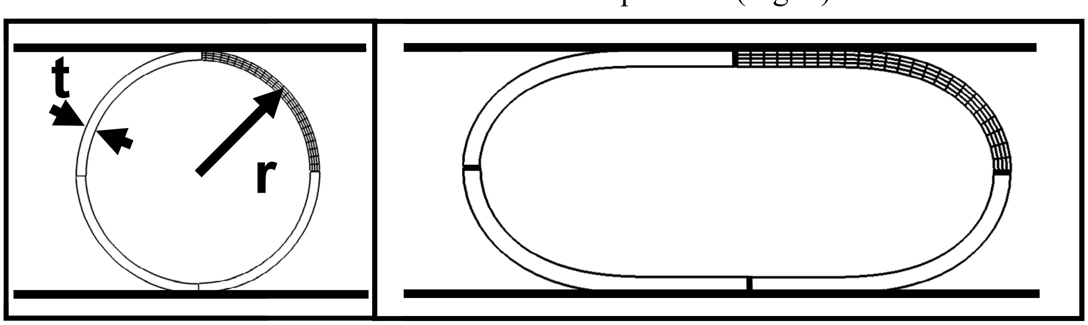


Figure 5. Undeformed and crushed pipe.

with 24 elements along the arc, four elements through the thickness, and one element through the length. Nodal constraints were used for the plane-strain and quarter-symmetry boundary conditions and the problem was driven by prescribing a 50 mm displacement to the rigid body. Contact between the anvil and pipe was enforced using an augmented-Lagrangian routine. A St. Venant-Kirchhoff elastic material (E = 186 GPa, v = 0.3) was used for this problem. The rigid body reaction force versus displacement predicted by FEBio was compared to the results from NIKE3D and ABAQUS.

The variations in the results predicted by the three FE codes (Fig. 6) was due to the differences in the linear elastic materials invariant to large rotations implemented in each FE code. Still, at peak displacement there was <3% difference in the predicted forces.

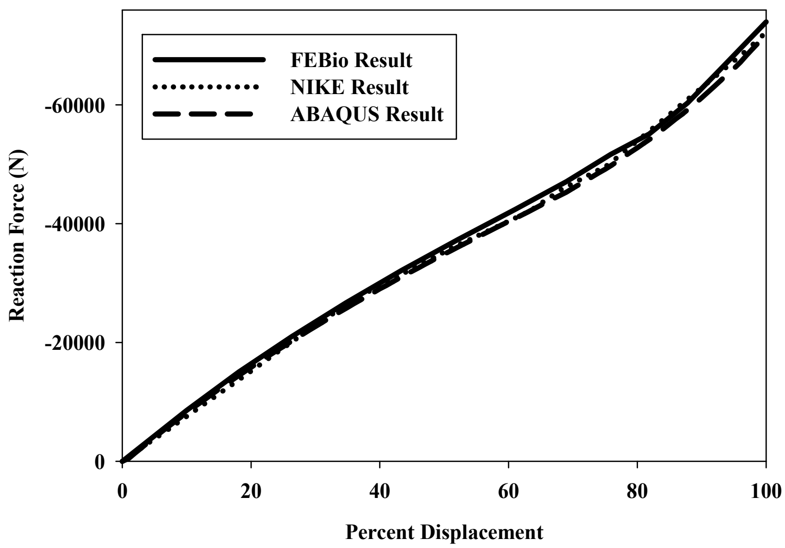


Figure 6. Anvil reaction force versus percent displacement results for FEBio, ABAQUS, and NIKE3D.

Cartilage Layer Compressed by a Flat, Rigid, Impermeable Surface. A half-symmetry finite element frictionless contact analysis was performed between a spherical elastic layer anchored to a rigid impermeable substrate and a flat impermeable rigid surface (Fig 7). The geometry was representative of the articular layer of an immature bovine humeral head, with a cartilage surface radius of 46.3 mm and a cartilage layer thickness of 0.8 mm. The converged mesh consisted of 20 trilinear hexahedral elements through the thickness, 50 along the radial direction, and 14 along the circumferential direction. The top of the cartilage layer was constrained by defining the top layer of nodes as a rigid body with all 6 DOF constrained. The deformation at the center of the articular layer was set to 0.095 mm (~12% of the thickness) by displacing the bottom nodes using nodal prescribed displacements. The cartilage layer was represented by the incompressible neo-Hookean constitutive model in Eq. (1) to approximate the short-time response of the tissue. The radial and circumferential stresses predicted by FEBio through the thickness of the middle of the cartilage layer were compared to the results from ABAQUS and NIKE3D.

The predicted radial and circumferential stresses through the thickness of the middle of the cartilage layer were the same for FEBio and NIKE3D, but the stresses predicted by ABAQUS were consistently slightly larger (Fig. 8). For this problem the small differences (<3%) are most likely due to FEBio and NIKE3D using an augmented-Lagrangian routine to enforce the material incompressibility and ABAQUS using a Lagrangian-multiplier routine.

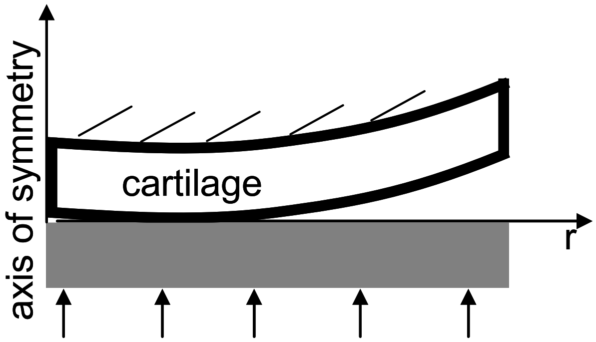
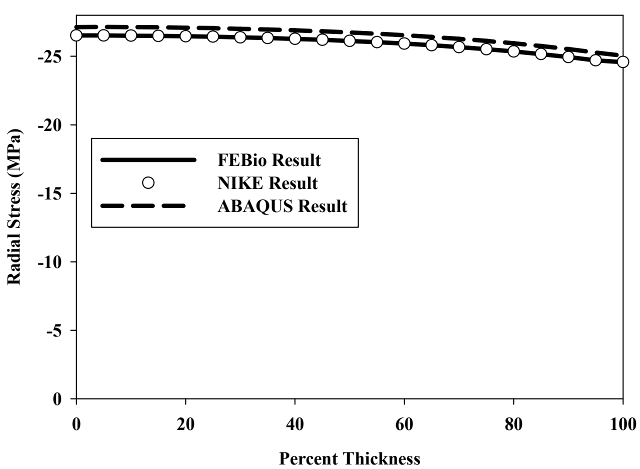


Figure 7. Cartilage compression model loading and boundary conditions.



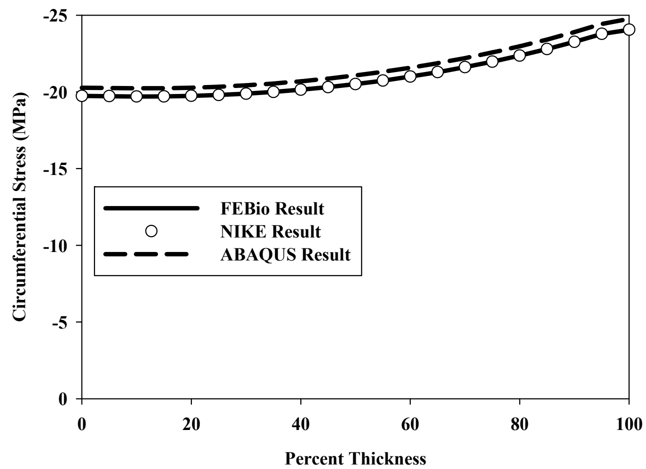


Figure 8. Cartilage radial stress (left) and circumferential stress (right) versus percent thickness results for FEBio, ABAQUS, and NIKE3D.

Conclusions

FEBio provides analysis methods and constitutive models that are relevant for this particular field and in this regard differ from many of the existing software tools that are currently being used by researchers. But, before FEBio could be trusted to solve more complex problems it needed to be verified with solutions from established codes and the results from FEBio for these benchmark problems were nearly identical to the solutions produced by ABAQUS and NIKE3D. This excellent agreement should give researchers confidence going into a new research problem that FEBio can provide accurate results. Still, problem specific verification (mesh convergence studies) and validation (comparison to a "Gold Standard") are necessary to assure that the predictions from FEBio are accurate for a specific problem.

Acknowledgments
Financial support from NIH #R01GM083925 and #R01AR047369 is gratefully acknowledged.

References

1. Davids, N. and M.K. Mani, *A finite element analysis of endothelial shear stress for pulsatile blood flow.* Biorheology, 1974. **11**(2): p. 137-47.
2. Doyle, J.M. and P.B. Dobrin, *Finite deformation analysis of the relaxed and contracted dog carotid artery.* Microvasc Res, 1971. **3**(4): p. 400-15.
3. Janz, R.F. and A.F. Grimm, *Finite-element model for the mechanical behavior of the left ventricle. Prediction of deformation in the potassium-arrested rat heart.* Circ Res, 1972. **30**(2): p. 244-52.
4. Matthews, F.L. and J.B. West, *Finite element displacement analysis of a lung.* J Biomech, 1972. **5**(6): p. 591-600.
5. Farah, J.W., R.G. Craig, and D.L. Sikarskie, *Photoelastic and finite element stress analysis of a restored axisymmetric first molar.* J Biomech, 1973. **6**(5): p. 511-20.
6. Belytschko, T., et al., *Finite element stress analysis of an intervertebral disc.* J Biomech, 1974. **7**(3): p. 277-85.
7. Maas, S.A. and J.A. Weiss. *FEBIO User's Manual* 2007 [cited; Available from: http://mrl.sci.utah.edu/software.php.
8. ASME, *Guide for verification and validation in computational solid mechanics*. 2006, American Society of Mechanical Engineers.
9. Babuska, I. and J.T. Oden, *Verification and Validation in Computational Engineering and Science: Basic Concepts.* Computer Methods in Applied Mechanics and Engineering, 2004. **193**(36-38): p. 4057-4066.
10. Oden, J.T., *Finite Elements of Nonlinear Continua*. 1972: McGraw-Hill, New York.
11. Treloar, L.R.G., *Stress-Strain Data for Vulcanised Rubber Under Various Types of Deformation.* Trans Faraday Soc, 1944. **40**: p. 59-70.