Finite Element Modeling in Musculoskeletal Biomechanics

Thomas D. Brown
University of Iowa

Numerical approximation of the solutions to continuum mechanics boundary value problems, by means of finite element analysis, has proven to be of incalculable benefit to the field of musculoskeletal biomechanics. This article briefly outlines the conceptual basis of finite element analysis and discusses a number of the key technical considerations involved, specifically from the standpoint of effective modeling of musculoskeletal structures. The process of conceiving, developing, validating, parametrically exercising, and interpreting the results of musculoskeletal finite element models is described. Pertinent case study examples are presented from two series of finite element models, one involving total hip implant dislocation and the other involving femoral head osteonecrosis.

Key Words: stress analysis, computations, tissue mechanics

Transmission of mechanical load is a salient function of musculoskeletal tissues, organs, and prostheses. Purposeful musculoskeletal activity depends both on local availability of the necessary mechanical impetus and on capability of the involved tissues/ organs/ implants to sustain the necessary loads without undergoing structural failure. Functionally, structural failure can take one of several forms. There can be direct irreversible degradation of mechanical competence, such as from catastrophic rupture of a tendon, fatigue fracture of a bone, plastic yield of a surgical implant, etc. Or there can be deformations which, while reversible, are so excessive (e.g., hyperlaxity of an articular joint) as to compromise neuromuscular control, load actuation efficacy, or global structural stability. Alternatively, failure can occur because of a biologically mediated cascade of events accompanying disruption of tissue homeostasis, due to the local mechanical milieu being outside of physiologically appropriate limits. Examples of the latter include disuse osteoporosis of a bone, or catabolic degradation of articular cartilage matrix by chondrocytes experiencing excessive geometric distortion. Successful transmission of necessary mechanical actuation therefore depends on avoiding inappropriate intensity of mechanical loading. For that reason, ascertaining levels of local mechanical stress is one of the central problems of biomechanics.
Background/Historical Overview

Much of the archival literature in the field of engineering structural mechanics, prior to about the 1970s, dealt with using analytical mathematical techniques to determine stress distributions in objects of engineering interest. The theoretical basis for determining mechanical stress in a continuous isotropic elastic medium dates from Cauchy’s work in the 1820s (Sokolnikoff, 1956). This takes the form of partial differential equations for equilibrium (Equ. 1) and for geometrical compatibility (i.e., no void openings or material overlaps) (Equ. 2), subject to a set of boundary conditions such as a prescribed traction force distribution (Equ. 3):

\[ \tau_{ij,j} + F_i = 0 \quad (i, j = 1, 2, 3), \]

\[ \nabla^2 \tau_{ij} + \frac{1}{1 + v} \Theta_{,ij} = -\frac{v}{1 - v} \delta_{ij} \text{div} \vec{F} - (F_{i,j} + F_{j,i}), \]

and

\[ \tau_{ij} \vec{e}_j = \bar{T}_i. \]

Here, per indicial notation convention, the subscripts \(i, j, k\) denote the respective orthogonal directions, repeated subscripts denote summation over the range of 1 to 3, and subscripted commas designate partial differentiation. The symbol \(\tau\) denotes stress, \(\vec{F}\) designates the body force vector (whose components are \(F_i\)), \(\nabla^2\) is the Laplacian operator (\(\nabla^2 \phi = \phi_{ii}\)), \(v\) is the Poisson ratio, \(\Theta\) is the first stress invariant (\(= \tau_{ii}\)), \(\delta_{ij}\) is the Kroniker delta (\(\delta_{ij} = 1\) if \(i = j\), 0 otherwise), div is the divergence operator, \((\text{div} \vec{A} = A_{i,i})\), \(\vec{e}_j\) are the direction cosines of the external surface normal \(\vec{e}\), and \(\bar{T}\) is the (traction) stress vector acting at a surface site whose external normal is \(\vec{e}\). Solving such sets of partial differential equations called for a high level of skill in applied mathematics, with many classes of stress analysis problems involving extraordinary solution complexity. Even so, analytical solutions were generally able to address only objects of simple geometry, comprising materials with relatively simple mechanical properties, subjected to relatively simple loading conditions. Since biological systems are geometrically irregular, and since they generally have complicated distributions of material mechanical properties and are subject to complicated loadings, analytical solutions from classical mechanics were seldom either feasible or meaningful for stress analysis problems in biomechanics.

That situation has changed radically over the last three decades, due to the advent of numerical approximation techniques, enabled by digital computation. Finite element analysis (FEA) has arguably been the flagship among these approximation techniques. In FEA, the basic idea is that rather than obtaining an exact algebraic solution of the governing partial differential equations throughout the domain of interest, one instead numerically solves a system of simultaneous equations that arise from enforcing those governing equations for an array of discrete simplified subdomains (elements). Within these individual elements, specific interpolation functions (usually polynomials) are assumed, from which continuous internal variables (e.g., strains) are piecewise approximated on the basis of corresponding parameters (e.g., displacements, in the case of strain) evaluated at a discrete number of characteristic local points (nodes).

This conceptual approach had its mathematical roots in Courant’s work in the early 1940s (Bathe, 1996), with the first applications in physics (Synge, 1957).
and in engineering (Argyris & Kelsey, 1954) appearing in the mid-1950s. Coinage of the term “finite element analysis” is frequently attributed to R.W. Clough (1960), who used it in a civil engineering conference presentation in 1960. The ensuing decade witnessed vigorous development both of the underlying theoretical basis and of its computational implementation, driven especially by analysts in the fields of structural mechanics and aeronautical engineering. Beyond its initial application to static structural problems, the FEA piecewise approximation approach proved applicable to dynamic structural problems, to fluid mechanics, to thermal and mass transport analysis, and to electromagnetic field analysis and other continuum field problems.

The Role of FEA in Biomechanics

The first applications of FEA to biomechanics, which appeared in 1972, involved isotropic linearly elastic structural analyses of bone, performed by groups working independently in the U.S. (Rybicki et al., 1972) and in the Netherlands (Brekelsmans et al., 1972). This new capability’s benefit to the field of biomechanics was immediately evident: For the first time, one could calculate mechanical stresses in bodies having complicated shapes, complicated material distributions, and/or complicated loadings. Other types of biomechanical complexity subsequently addressed have included material anisotropy, material and geometrical nonlinearity, contact and interface nonlinearities, time-variant loadings, adaptive behavior (material and geometrical), and fluid/structure interactions. The FEA biomechanics literature now encompasses many hundreds of papers. Surveys of work of the first (Huiskes & Chao, 1983) and second (Huiskes & Hollister, 1993) decades in this field were necessarily restricted to reviewing just small subsets of the highlights.

The purpose of the present article is to discuss a number of considerations pertinent to applying FEA to research problems in biomechanics, especially in musculoskeletal structural analysis. Didactic presentation of the formulative basis for FEA, normally the subject of entire semester-long university courses at the advanced engineering undergraduate or graduate level, is necessarily beyond the scope of this brief discussion.

Biomechanical FEA studies sometimes involve man-made geometries (e.g., surgical implants), sometimes natural geometries (anatomical structures), and sometimes both. For man-made geometries, mesh zonings at the simplest level can be based on nominally corresponding geometrical abstractions. More typically nowadays, they are user-built to approximate the physical objects at an appropriate level of detail, or they are generated automatically or semi-automatically by utility routines which translate the data in the computer aided engineering files used to actually manufacture the part. Regarding natural geometries, a key consideration is whether the anatomic representation is generic vs. case- (or patient-) specific. Source geometric data for the meshing of natural structures nowadays usually come from three-dimensional imaging modalities such as computed axial tomography (CT), magnetic resonance imaging (MRI), or confocal microscopy. Sometimes photographs of serial physical sections are used instead, and on rare occasions, plane film or biplanar xrays. Geometric fidelity typically depends on the resolution of the pixel or voxel image source. Another consideration is that such image sources often have limited capability for delineating subregions of differing material behavior.
Finite element program libraries now incorporate a wide range of options for material property representation, including linear and nonlinear elasticity, plasticity, hyperelasticity (rubber-like materials), viscoelasticity, poroelasticity, and bimodal behavior for tension vs. compression. Various types of material symmetry are readily accommodated, including isotropy, transverse isotropy, and orthotropy. Since material representation is normally specified in a piecewise manner, the presence of material inhomogeneity presents no computational difficulty. Commonly used treatments include regional homogeneity, element-specified properties, and (less frequently) element properties interpolated from node-specified properties. Computational treatments also are available for many commonly used composite arrangements. For practical purposes, the realism of material representation in biomechanical problems is far more frequently limited by available experimental data than by algorithmic capabilities.

There are two primary sources of complexity in the loadings typically encountered in biomechanical finite element studies. The first complexity is temporal: Some systems are appropriately analyzed with a single static loading, other systems with a series of different quasi-static loadings, yet other systems merit fully dynamic treatment (i.e., inertial effects included). Occasionally, modal analysis is invoked for applications such as vibration studies. The second source of complexity is the spatial distribution of loading. Point loadings (the simplest situation) are relatively rare, as are line loadings. Most frequently, loadings of interest occur across surfaces, and most commonly they are nonuniformly distributed. These can take the form of prescribed traction distributions, or they can be loading-continuous, as for example in contact problems.

Sometimes it is appropriate to also superimpose a distributed body force to account for weight. Besides loading, the other commonly encountered type of external influence is displacement boundary conditions. These can take the form of a constraint against motion in one or more degrees of freedom, either applied at individual element nodes or along element edges or faces. Alternatively, rather than fixities, displacement boundary conditions can involve prescribed nonzero displacements in one or more degrees of freedom, again specified at nodes, along element edges, or over element faces. For some problems it is appropriate to make provision for mechanical interaction across an interface. One type of interface treatment is friction-dependent slip/nonslip. Another is gap closure: no tension transmission, but allowing for transmission of compression at sites of gap closure. Formal contact treatments can include shear as well as compression, and in increasing order of complexity can involve sticky contact (no relative tangential motion of contact surfaces), small slip contact, or large displacement sliding contact. And, a number of specialty formulations have been developed to model other types of interface behavior.

**Stages of the Modeling Process**

Finite element studies generally involve five stages. The first stage is development of specialty coding, if appropriate to the problem at hand. (This step is now unnecessary in most studies, which are well accommodated by the many features and capabilities already incorporated in the major program packages available commercially.) The next phase, known as preprocessing, involves prescribing the mesh geometry, specifying the material property distributions, and designating the load-
ing. Then the finite element solution is executed computationally, a process usually involving little or no direct interaction with the user. There then follows a stage known as postprocessing, where the FE solution’s raw output is used to compute variables of interest, and where selected information is displayed graphically. The final stage is that of interpretation of the results, a process heavily dependent on analyst judgment.

In research settings, these four (or five) stages typically proceed iteratively, over weeks or months, as different aspects of a model are brought into ever-better consonance with physical/clinical reality, and as issues influencing system behavior are explored in context. Computer CPU time to execute the actual finite element solutions (Stage 3) is usually just a small fraction of elapsed calendar time in an FE project, although models at the more complex end of the spectrum can sometimes involve multihour, overnight, or even multiday solution runs.

The great majority of biomechanical finite element models utilize commercial software, both for pre- and postprocessing (e.g., PATRAN, SUPERTAB, FEMGEN, GIFTS) and for the actual solution phase (e.g., ABAQUS, ANSYS, NASTRAN, MARC, ADINA). These are all large program packages with extensive libraries of features from which the user can select. In recent years the major FE solution programs have greatly improved their qualities and capabilities regarding user interface, and thus have gained an increased marketplace share in pre- and postprocessing.

### Elements and Nodal Forces

To appreciate why the field of finite element analysis is so intimately coupled with digital computation, it is useful to understand how the finite element approach to piecewise discretization is implemented numerically. Here the situation of elastic structural analysis, the simplest and most common of FE formulations, is illustrative. Consider the situation of a linear spring (Figure 1a), which can alternatively be viewed as a one-dimensional finite element. Application of a force \( f \) causes the two ends of the spring (or nodes of the element) to displace a distance \( u \), depending on the stiffness \( k \) of the spring. Mathematically, this relationship is \( f = k \cdot u \). The two-dimensional counterpart (Figure 1b) is a quadrilateral element. Here there are four nodes, each of whose displacements involves two variables (e.g., an \( x \)- and a \( y \)-component, in conventional Cartesian coordinates.) Hence the element’s displacement is characterized in terms of an eight-member nodal displacement vector, \( \{u\} \). Similarly, each node can experience a distinct force, again involving two components, so the system’s overall force can be expressed in terms of an eight-member nodal force vector \( \{f\} \).

The set of nodal displacements resulting from a given set of nodal forces (Equation 1b) depends on the element stiffness, specified in terms of a two-dimensional array \([k]\) known as the element’s stiffness matrix. Mathematically, analogous to the spring, this relationship is \( \{f\} = [k] \cdot \{u\} \). Taking this idea further, consider an assemblage of two-dimensional elements (Figure 1c). Most of the nodes in this system are shared by two or more elements, which provides a mechanism through which the deformations of all elements are interlinked. Mathematically, again analogous to the spring, the global behavior of this system is conveniently described in terms of the vector/matrix relationship \( \{F\} = [K] \cdot \{U\} \), where \( \{F\} \) is the global force vector (40 members, in this 20-node, two-dimensional system), \( \{U\} \) is the
Figure 1 — Elements and stiffness matrices. For a simple linear spring (1a), essentially a one-dimensional element, application of a force $f$ causes a displacement $u$, in inverse proportion to the spring stiffness, i.e., $f = k \cdot u$. In the 2-D counterpart (1b), the set of nodal displacements $\{u\}$ and forces $\{f\}$ are interrelated in accordance with a geometrically and materially dependent local stiffness matrix $[k]$: $\{f\} = [k] \cdot \{u\}$. Similar concepts apply in a multi-element system (1c), where the forces $\{F\}$ and displacements $\{U\}$ of the nodes of the entire system are linked in terms of the relation $\{F\} = [K] \cdot \{U\}$, where the global stiffness matrix $[K]$ is constructed on the basis of individual nodes’ associations with multiple elements.

global displacement vector (again, 40 members), and $[K]$ is a $40 \times 40$ array known as the global stiffness matrix. For a given $\{F\}$, calculating the corresponding $\{U\}$ requires solving a system of 40 equations with 40 unknowns.

The coefficients in the local stiffness matrix depend on two factors, one physical and the other geometric. The physical factor is the mechanical behavior of the material which the element comprises. In two dimensions, this can be characterized as a $3 \times 3$ coefficient array $[D]$ linking local stress (a three-member vector, two of whose members are the in-plane normal stress components, and one is the in-plane shear component) and local strain (similarly, three components). The geometrical factor is a relationship—usually an interpolation polynomial—through which the element’s local internal strain is assumed/defined to depend on nodal displacements: this also takes the form of a two-dimensional array $[B]$. The local stiffness matrix $[k]$ is obtained by element-wide integration (Equation 2) of the kernel $[B]^T [D] [B]$, where the superscript $T$ denotes transposition. Element-wide integration involves an area integral in two dimensions, and analogously a volume integral in three dimensions.

Appropriate material property representation, the $D$ matrix, is often problematic in biomechanical FE models. In three-dimensional generality, stresses and strains at a point in space each involve three components, therefore requiring a set of 36 coefficients to fully specify the interrelationship. For linear elasticity, this is known as Hooke’s Law: $\{\sigma\} = [C] \cdot \{\varepsilon\}$, again a vector/matrix relationship:
Although the stiffness matrix is symmetrical about its main diagonal (i.e.,
only 21 of the 36 members are unique), virtually never in biomechanics is this full
set of anisotropic material coefficients available. Instead, varying levels of simpli-
ification are invoked, to fill out the array on the basis of limited available information. For example, Cowin and Sadegh (1991) have characterized cortical bone’s behavior (units of GPa) as orthotropic:

\[
\begin{bmatrix}
\sigma_{11} \\
\sigma_{22} \\
\sigma_{33} \\
\sigma_{23} \\
\sigma_{13} \\
\sigma_{12}
\end{bmatrix} =
\begin{bmatrix}
C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\
C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\
C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\
C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\
C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\
C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66}
\end{bmatrix} \begin{bmatrix}
e_{11} \\
e_{22} \\
e_{33} \\
e_{23} \\
e_{13} \\
e_{12}
\end{bmatrix}.
\] (4)

Most nodes in a global structure are local nodes of multiple elements within
that structure. Since mechanical equilibrium at these (shared) nodes therefore de-
pends on the local stiffness matrices of all elements of which they are a part, there
is a basis for assembling all the local element stiffness matrices into a global stiff-
ness matrix. Owing to the multiplicity of terms in the local element stiffness matri-
ces, plus the large number of individual elements needed to discretize a complex
global anatomic structure such as a bone, most contemporary finite element mod-
els in biomechanics involve a very large global stiffness matrix, often with dimen-
sions in the hundreds of thousands or more. One noteworthy recent model of an
entire femur, zoned from micro-CT data, involved over 96 million three-dimen-
sional elements (van Rietbergen et al., 2003). Fortunately for computation pur-
poses, individual nodes are shared by elements only in a “nearby” neighborhood
of the overall global structure. If the nodes and elements are sequentially num-
bered appropriately (built-in optimization routines for such purpose are now stan-
dard in commercial FE programs and operate transparently to the user), global
stiffness matrices tend to be very sparse, with their nonzero terms clustered near
the main diagonal.

This sparseness greatly facilitates specialized simultaneous equation pro-
cessing algorithms that obtain solutions with orders of magnitude of fewer alge-
braic operations (i.e., orders of magnitude more quickly) than “brute-force” matrix
inversion. Even so, FE computations have always demanded high processor speed,
large RAM, and large amounts of disk storage. Finite element analysis initially
was exclusively the province of mainframe computers, but continued advances in
platform capabilities has enabled continued decentralization, first to high-end
engineering workstations and nowadays routinely with PCs. Of course the larger
and faster the platform, the better. Powerful computers especially hold important advantages when working with nonlinear models, where the simultaneous equation system solutions need to be obtained many times, rather than just once as is the case in linear problems.

In structural problems, by far the most common class of FE formulations currently used in biomechanics, the base variable that emerges from equation system processing is the set of displacements of the nodes. (Recall, one is simply solving for \( \{U\} \) from the equation \( \{F\} = [K] \cdot \{U\} \)). Normally, however, information about site-specific point displacements in a structure is of limited practical utility. Rather, one is usually interested in strains and/or stresses. To get that information from the base dataset, the global node displacements are referenced back to the corresponding local nodes of individual elements and the strain displacement ([B]) matrices of those elements, thus allowing evaluation of strain. Going further, the strains are combined with the elements’ material coefficient ([C]) matrices, for evaluation of stress. Going still further, stresses and strains are typically used to evaluate yet other quantities of physical or physiological interest, such as principal stresses, strain energy density, von Mises equivalent stress, etc. In commercial programs, such postprocessing calculations normally are transparent to the user, who needs merely to select from an extensive menu of output data options.

**Nonlinear Problems**

The situation is complicated somewhat when structures undergo relatively large deformations, such that deformation-dependent definitions of strain (e.g., Green-Lagrange strain) and stress (e.g., Cauchy stress) are necessary (Bathe, 1996), or when materials have deformation-dependent mechanical properties. Such nonlinear situations require that the finite element problem be solved incrementally. To implement this, a small fraction of the total load is initially applied, such that the stiffness matrix can be reasonably approximated based on the undeformed geometry, or the undeformed material properties, and a nodal displacement solution obtained. The deformed nodal displacements are then used to update the geometry, or the material properties, at which point an updated stiffness matrix is calculated, prior to application of an additional increment of the total applied loading, and so forth. Since the loading increments used for this purpose need to be relatively small, this vastly increases the computational effort compared to that required for linear problems.

One of two major incremental solution strategies is utilized for nonlinear problems. One of these is the explicit formulation, which is unconditionally stable numerically, and which involves marching the process monotonically forward at prespecified increments which are sufficiently small that incremental adjustments of the nodal coordinates are not required to propagate through the structure at speeds greater than the physically admissible (i.e., sonic) deformation wave velocity. Alternatively, and more commonly, a conditionally and numerically stable implicit solution technique is employed, wherein a new solution is extrapolated from the previous increment, and nodal displacements are iteratively adjusted until mechanical equilibrium is re-achieved within some specified tolerance limit. Depending on the specifics of the situation, it may or may not be possible to implicitly converge to a new equilibrium configuration for the increment size chosen. If not, implicit solvers typically cut back the increment size and repeat the iterative
procedure, recursively attempting to find a new equilibrium configuration involving lesser extrapolation from the previous increment’s equilibrium solution, with a solution error abort condition being invoked if the increment size shrinks below a specified threshold.

Zoning

In developing biomechanical FE models, arguably the most critical and challenging aspect is zoning the physical structure into elements. Unless zoning is done appropriately, reliable stress/strain data will not emerge. Many types and subtypes of elements are available in program libraries. The most commonly utilized element family, known as full continuum elements, imposes relationships between element nodal forces and nodal displacements, in such a manner as to account for all stress and strain components present in the corresponding solid continuum. Some examples are shown in Figure 2. Depending on the number and positions of local nodes in an element, varying levels of accuracy and refinement can be achieved in approximating the spatial distribution of stresses and strains in the corresponding domain of the actual physical solid.

A key consideration in biomechanics is that using higher-order elements (i.e., those with more nodes than the number of vertices in the corresponding geometric solid) allows generating elements whose edges or faces are curved. Also, to deal with the geometrical transitions that are pervasive in many biological structures, it is often convenient to specify that certain pairs (or even groups) of nodes in continuum elements remain coincident, thus “degenerating” the parent solid.

![Figure 2 — Typical 3-D elements. Higher-order elements (which have additional nodes at locations other than the element vertices, such as at intermediate positions along edges or vertices) are capable of more complex performance, such as registering heterogeneous distributions of stress and strain, and/or mapping onto curved boundaries of the object being meshed.](image)
Finite Element Modeling 345

into a different type of solid having fewer vertices/edges/faces (Figure 3). Other families of elements are designed to offer economies of simplification, by not computing certain components of stress or strain for situations where those components either are negligible or are of no physical interest. Examples are truss and spring elements (Figure 4), beam elements (Figure 5), and shell elements (Figure 6). Yet other families of elements implement special considerations that bear upon the interrelationship of nodal forces and nodal displacement, such as contact elements (Figure 7) or crack tip elements (Figure 8).

Two other concepts of element definitions merit mention, in light of frequent biomechanical implications. The first is local coordinates. Individual elements are always specified in terms of their local nodes, which follow a specific numbering sequence (Figure 9). The local node numbering sequence in turn provides a basis for establishing local coordinate directions, specific to each element, and not necessarily coincident with global coordinate axes. Local coordinate directions are useful in biomechanics because they provide a basis for incorporating local tissue structure, for example in terms of direction-dependent mechanical properties (anisotropy), or in terms of stresses or strains relative to tissue planes. Cortical bone, for example, is approximately half again as stiff in the prevailing direction of its secondary osteons (e.g., longitudinally, in a diaphysis) than transversely, a

Figure 3 — Transition of element subdivisions in physical domains can be accomplished using “degenerative” elements, wherein multiple nodes of parent elements with a higher number of surfaces are defined to be coincident. Transitions of this type are often helpful when meshing geometrically complex biological structures.
Figure 4 — Spring or truss elements are often used to represent structures that transmit only unidirectional force. Common examples in musculoskeletal biomechanics are ligaments and tendons.

Figure 5 — Beam elements are used to represent slender prismatic structures capable of transmitting both forces and moments. The local stiffness matrix depends on the type of cross-sectional geometry in the local coordinate system \((R,S,T)\), which the user identifies from a menu of options in the program’s element library. The user then supplies the pertinent geometrical data \((H_1, H_2, \text{etc.})\) needed to fully prescribe that particular cross-section.

factor which when incorporated in FE material property input leads to computed stress fields that are strikingly different from those computed when tissue material directionality is ignored (Ricos et al., 1996). A second zoning concept, pertinent especially to inferring stresses at surfaces or interfaces, is that continuum elements, strictly speaking, can evaluate “exact” stresses only at a discrete small number of internal points— termed integration points or Gauss point—rather than everywhere. The specific number of Gauss points, and their local coordinates, depends on the
Figure 6 — Shell elements are useful to represent thin material layers, such as the
subchondral plate in juxta-articular stress analysis problems. Normally the accuracy of
finite element results in continuum regions degrades when element aspect ratios (ratio of
longest to shortest sides) depart excessively from unity. With shell elements, at the (usually
inconsequential) expense of assuming a specific distribution of displacement, stress, and
strain in the direction of the element thickness (here, the local coordinate direction t), one
avoids the need to zone elements with in-layer (r,s) dimensions comparable to the thickness
dimension.

Figure 7 — Contact elements account for variable interaction between a pair of material
surfaces. They typically evaluate the nodal contact forces needed to preclude interpenetration
of solid bodies. Since the specific regions of the respective surfaces coming into contact
depend on the resultant force between the two bodies, the associated global stiffness matrix
necessarily changes accordingly. This is an example of a geometric nonlinearity, requiring
multiple solutions to arrive at the system’s equilibrium configuration if an implicit equation
solution formulation is employed.
Figure 8 — Crack-tip elements are specialized higher-order elements having nodes positioned at specific fractional distances (1/4) along edges bordering a propagating crack. The corresponding strain-displacement matrix [B], relating the element’s internal strain distribution to its set of nodal displacements, allows such elements to reproduce the stress/strain singularity (i.e., local increase to infinity) ideally present at the tip of a sharp crack.

Figure 9 — Specified ordering of the nodes for each element allows definition of a local coordinate system (r,s,t), often useful for purposes such as prescribing tissue-structure-dependent material anisotropy.
particular formulation equations for each type of element (Figure 10). When stress values are needed elsewhere—commonly, for display purposes, on the outward-facing surfaces of elements on the exterior of a structure—by necessity these are simply outward extrapolations of stresses in nearby underlying Gauss points. At sites of high stress gradient, this can be an important caveat in interpreting stress results.

Practical use of finite element analysis in the field of biomechanics involves several hurdles that are seldom so problematic in traditional areas of stress analysis. First, commercially available FE programs, while ever more powerful and versatile, are necessarily developed as profit-making software products targeted to market demand. The vast majority of that market demand is for engineering analysis and design of nonbiological systems. In production stress analysis environments, highly efficient code performance can be extremely important, whereas data input/output is typically fairly straightforward. For biomechanical FE studies, which typically occur in a research environment, the priorities tend to be different: data input/output often is tedious or burdensome, whereas the numbers of cases needing to be run (e.g., to test a hypothesis or to document a phenomenon) can be relatively modest. Quite often, the “canned” capabilities of commercial software need to be augmented by user-written utility routines, to address special considerations involved in biomechanical studies.
Specialty biomechanics augments unfortunately tend to be of little interest from a theoretical or numerical analysis perspective, and they tend to serve only a very small segment of the marketplace. FE users in the biomechanics community therefore share a stake in nurturing such developments wherever possible. Another concern, also of a logistical nature, is that updates of commercial software, which tend to be market-driven, are rapid, relentless, and for practical purposes mandatory. The majority of that software market is in private industry, where organizational cultures have evolved to provide their R&D component with the resources needed to remain competitive. Resource availability tends to be less in most computational settings engaged in biomechanical modeling, such as universities or health care research institutes.

Effective use of FEA in biomechanics also has several unique scientific considerations. First, the analyst should appreciate that individuals with a clinical or biological background are often skeptical about the validity and utility of mathematical models. Thus it is important to clearly justify why computed stresses help answer the question at hand. Where possible, avoidance of complicated parameters or indices helps keep that basic message comprehensible and compelling to nonspecialists. Input data (geometry, materials, loadings) should be based on a situation as closely related as possible to the problem at hand, and those data should be as physically and biologically grounded as is practical. FE model development in new areas therefore often requires that the analyst be prepared, either directly or by collaboration, to collect specialty input data not available in the literature.

Geometry

Regarding geometry, three-dimensional models now constitute the norm. About the only appropriate uses of two-dimensional formulations are when the system of interest possesses approximate geometrical and material symmetry about a single axis (axisymmetric simplification), when it is a thin surface such as a membrane (two-dimensional plane stress), or when it is a cross-section of a long, prismatic structure in which all sections can be assumed to behave comparably (two-dimensional plane strain). Two-dimensional idealizations also are appropriate, as an intermediate stage of development, for formulations having an appreciable degree of conceptual novelty. Otherwise, however, two-dimensional FE models appear relatively infrequently in contemporary biomechanics research. Digitized image sources are generally preferable, for purposes of accurately capturing the complex geometry of biological constructs. Where possible, these should be coupled with objective delineation of internal material boundaries or interfaces. Oftentimes, interpretation of computational results is facilitated by a graphic demonstrating FE mesh overlay onto the source geometry. One situation where idealized geometry is appropriate, however, is with man-made objects such as surgical implants; there the best source of information is data from the CAE files used for the actual manufacturing.

Material Properties

Because of biologic variability, biomechanical constructs almost always involve some degree of idealization of material properties. Judgments in that regard are within the analyst’s legitimate purview, and can and should be made in a manner that facilitates the overall objectives of the analysis. Since material properties are
seldom unambiguously known in biomechanical FEA, it is usually a good practice to conduct preliminary sensitivity trials, to gage solution sensitivity to plausible variations in the material input. This is particularly the case in nonlinear problems, where the “character” of the solution can vary appreciably, depending on where one is operating in the state space of system parameters. Meshing should be done in such a manner that all distinct material regions are represented. In that regard, for a given problem size (i.e., for a given number of global nodes), low-order elements having relatively few local nodes are normally preferable to high-order elements for biomechanical models. The former provide better spatial resolution for delineating material property input, a consideration that usually far outweighs the latter’s slight intrinsic numerical performance superiority.

Loadings

Loadings are another aspect of biomechanical FE models that usually call for appreciable judgment on the part of the analyst. Most living systems experience wide fluctuations of loading during function, so it is usually desirable to input duty cycles that reflect this variation. For example, in studying stresses in a total hip prosthesis, it is desirable to consider a range of loadings that span those developed during the gait cycle. Computationally, this is easy to do in linear problems, since the global stiffness matrix need not be recalculated; commercial FE packages normally include an option for the user to specify multiple loading cases. Even in nonlinear problems, however, the additional credibility provided by considering a physiologically representative sequence of loadings normally more than justifies the extra time and expense involved. Most loadings in biomechanical systems are distributed, rather than concentrated. Finite element codes provide a menu of options for specifying those distributions. A commonly used choice is to specify loadings individually for each element on the surface of a loaded structure. Finally, in nonlinear problems, as with material property assumptions, it is usually prudent to evaluate solution sensitivity to loading assumptions, since model behavior can be qualitatively very different in different regions of the parameter space.

Post-Processing

In reporting results from biomechanical FE models, the analyst needs to be cautious about reporting trends that are specific to individual components of stress or strain. Different components of computed stress or strain can exhibit very different (sometimes even contradictory) responses to input variables. In that common situation, the analyst bears responsibility for choosing which, among many, output variable(s) merit attention, for purposes of characterizing the relevant aspects of system behavior. Sometimes, conventional multicomponent indices (e.g., von Mises equivalent stress) available from FE postprocessing programs can be very helpful in that regard, but their use should be limited to contexts in which the index has direct physical meaning. In other situations it may be appropriate to define and evaluate ad hoc multicomponent indices, a step normally requiring off-line calculations using a separately written program (e.g., in FORTRAN or C++) or a spreadsheet.

In biomechanics, good graphics are key to a high-impact finite element model. In discussing stress distributions in a biomechanical construct, it is almost always useful to first give the reader/audience the “lay of the land” by presenting at least
one full-field view of the structure being analyzed. Often a contour plot of the baseline stress distribution works well for that purpose. However, to maximize their quantitative utility, contour plots require clearly identifiable gradations, and their scale annotations typically need to be enlarged from plotting program defaults in order to be easily legible. Individual parametric influences are usually best illustrated by line plots, which afford space-economical demonstrations of the effects of univariate, bivariate, or even trivariate input changes. Again, however, while most finite element postprocessing programs include basic line-plotting capabilities, these often lack sufficient flexibility for use in biomechanical systems. Access to an adjunct imaging/plotting/analysis package is therefore highly desirable to enable construction of ad hoc graphics.

**Reliability and Validation**

Since FEA is by nature an approximation technique, it is important to demonstrate that the results obtained are reliable. One consideration is to show that the mesh is sufficiently refined that the solution will not change appreciably with additional mesh refinement, i.e., that the numerical solution has converged mathematically. Two variants of convergence testing are typically used in the general field of finite element analysis, h-convergence and p-convergence (Figure 11). P-convergence testing, which involves progressively higher order elements, is rarely performed in biomechanical models, because high-order elements are seldom appropriate, for reasons noted above. But even formal h-convergence tests are usually difficult to design in biomechanical modeling studies, since geometrically irregular structures seldom lend themselves to systematic progressive refinements of zoning. Instead, what is often done is to perform an “informal” h-convergence test, involving several alternative meshes with progressively increased resolution, but without the sort of uniformly systematic subdivisions that are practical for simple geometric shapes.

*Figure 11 — Successively refined spatial resolution can be achieved either by using progressively greater numbers of elements of the same order, or by progressively increasing the order of a fixed number of elements. The former is far more common in biomechanics, owing to the typical need to account for material heterogeneity. Documenting that the level of mesh refinement is such that the computed solution does not change appreciably for additional refinement is the province of convergence testing. Doing this for element number vs. element order is known as h- and p-convergence testing, respectively.*
Another broad approach to demonstrating FE solution reliability is to perform model validation. In order of increasing levels of model credibility provided, approaches to validation include:

1. Showing that the solution is reasonable and well-behaved, in the sense of being internally consistent;
2. Showing that the solution exhibits general consistency with existing literature;
3. Performing quantitative spot checks with existing literature;
4. Performing limit checks against available analytical solutions for related situations;
5. Comparing FE results with directly corresponding prospective experiments.

Depending on the system being studied, various sources of information may be appropriate for model validation. Surface strain measurements are one frequently used standard for comparison. Strain gages hold the advantage of giving direct numerical data. In biomechanical constructs, however, there can often be difficulty in making direct comparisons with FE, since the gage position and orientation need to be precisely and unambiguously registered relative to the numerical model. This is especially problematic in the all-too-common situation where the sites of primary interest in the model have high gradients of strain. Photoelastic coatings afford broader depiction of complex strain fields, but extracting specific numerical values is difficult and tedious, and often yield data which are at best only approximate. Normally, therefore, photoelastic recordings are useful mainly to confirm overall patterns in an FE-computed strain field.

An alternative basis for kinematic validation is to compare point-wise displacements or velocities. Sensors used for that purpose include both contacting modalities such as linearly variable displacement transformers (LVDTs) and extensometers, and noncontacting modalities such as eddy current transducers, ultrasound, and laser position sensors. As with strain gages, however, the problem of unambiguous spatial registration limits the available precision of comparisons. Finite element problems involving contact stress and/or fluid pressures are amenable to kinetic validation. Contact stresses can be measured statically with Fuji Film, or dynamically with sheet sensors such as TekScan® or EMed®. Fluid pressures can be measured using a wide variety of commercially available transducers.

Yet another approach to physical validation is to compare (integral) process variables, such as global accelerations or velocities of moving objects, flow rate of fluids, weight loss in wear simulations, or other such summary variables pertinent to the problem at hand.

**Interpretation**

A final key component to successful use of finite element modeling in biomechanics is appropriate interpretation of the model’s results. Doing this effectively requires that the FE analyst have at least a working knowledge of the physical/clinical/biological system under study. Since finite element models generate prodigious amounts of output data, it is important to avoid information overload. Direct knowledge of the subject matter is invaluable for identifying the FE phenomena that are most germane to the issues(s) under study. Subject knowledge also helps the analyst avoid making observations that will appear trivial or irrelevant to experts in
the field, and it helps the analyst identify contra-intuitive results that potentially merit reexamining the computational formulation or the input data. Subject knowledge also helps the analyst design parametric series that answer the question(s) at hand with reasonable economy of effort, and it provides a basis for making critical assessments of model limitations.

Interpreted in appropriate context, observations from biomechanical finite element models can be reported with a great deal of confidence and credibility. Numerical models are highly respected throughout the broad field of mechanics, and in many specialty areas their impact has been so pervasive as to displace experimentation as the preferred means of inquiry. This is certainly not yet the case in biomechanics, but toward that end, one can realistically look forward to continued dramatic improvements in the realism of biomechanical finite element models.

**Examples**

**Case Study 1**

Two sets of finite element case studies, drawn from recent research in the field of orthopaedic biomechanics, should serve to illustrate many of the above concepts. The first set of examples involves dislocation of total hip replacements. Here the general goal of finite element modeling is to gain direct mechanistic insight into ways to reduce the incidence of this troublesome surgical complication. Clinical registry data in this area unfortunately have many confounding variables, and thus have proven capable of documenting relatively few statistically significant cause/effect relationships. Direct laboratory experimentation, primarily with cadaveric material, is expensive, unwieldy, and can address only a narrow range of parametric influences and outcome measures.

For studying dislocations, the system of direct interest is the total hip implant, and the phenomenon of interest is disruption of normal ball-in-hemispherical-socket articulation, due to impingement of the ball’s support base (termed the neck) against the rim of the socket. The proximate cause of impingement is excessive angular motion of the hip, which tends to be associated with various untoward patient maneuvers, such as crossing one's leg or rising from a low toilet seat. The impingement site serves as a fulcrum, about which the femoral head ball tends to pivot, leading toward its being levered out of the socket.

Although geometrical idealizations were helpful in the exploratory phase of FE model development, this was a situation where credibility of the results in the eyes of clinicians strongly depended on working with familiar, realistic implant geometries. Therefore, the meshes (Figure 12) were zoned from CAE files (IGES files) used to manufacture the implants. A commercial pre-/postprocessing program (PATRAN) was used for that purpose. Since stresses in the metal femoral component and in the metal shell backing of the socket (acetabulum) were not of particular interest, and since the elastic modulus of the metal in those members was orders of magnitude higher than that of the plastic socket (made of ultra-high molecular weight polyethylene), the metal members were assumed rigid, affording their representation purely by a surface mesh. The polyethylene was treated as an elasto-plastic solid, zoned into low-order wedge (5-face) and hexahedral (6-face) continuum elements. Socket meshing resolution was concentrated at the expected impingement site, with suitable element density being inferred from h-type convergence testing (Figure 13). Because of expected stress concentrations at the
Figure 12 — Finite element mesh used to study dislocation of total hip replacement implants.

Figure 13 — Illustration of typical h-convergence testing, here undertaken to identify suitable meshing density of a total hip acetabular component in the zones for which femoral component neck impingement was feasible (30° zone) vs. nonfeasible (90° zone), respectively. These data showed that a problem size of at least 22,000 deg of freedom was necessary.
impingement site, implying potentially large strains (up to 10s of %) for the polyethylene, finite deformation material nonlinearity was invoked.

The problem was simultaneously driven by two types of input, both experimentally grounded. Kinematically, a sequence of triaxial rotations was input for the femoral component relative to the acetabular component, the latter’s outer (backing) shell surface being assumed to be rigidly fixed. Kinetically, a time-variant contact force was input at the center of the femoral head, based on joint resultant contact force calculated from an inverse dynamics multisegmental rigid body model run for (non-THA-implanted) individuals undergoing various motions known to be associated with dislocations in THA patients. Motion data for dislocation-risk activities were not previously available in the literature, and thus had to be captured specifically for the intended finite element studies (Nadzadi et al., 2003). The finite element model, run using the ABAQUS program, then computed internal and contact surface polyethylene stresses at each of typically 100 instants (frames) of discretized motion.

There were two outcome measures of primary interest: the range of motion prior to dislocation, and the peak moment developed to resist dislocation. Neither of these are standard menu items in commercial finite element programs. For range of motion, it was necessary to define a physically meaningful criterion to designate the exact instant of dislocation. The criterion used for that purpose was passage of the resultant contact force off the hemispherical bearing surface. This condition is associated physically with head escape from the socket, and was manifest computationally by the onset of numerical instability (i.e., Newtonian equilibrium could no longer be achieved.) Regarding peak resulting moment, special purpose FORTRAN postprocessing code was written to summate the resultant of all the individual moments (about the head center) of forces acting on external surface nodes of the polyethylene socket. The FE model was then experimentally validated using a laboratory bench-top fixture designed to induce dislocation due to imposed planar rotation of the acetabulum relative to the femur, and in which resisting moment could be measured by a load cell. This imposed planar rotation constituted a well-prescribed special case input sequence which could be unambiguously replicated computationally (Figure 14).

Once developed and validated, this finite element model was used to parametrically study dislocation propensity’s dependence on a wide range of influence factors. Those factors included variations in specific attributes of implant design, of surgical positioning, and of patients’ hip joint motions. Such parametric data lend themselves well to presentation in line plot format, using various graphics structures (Figure 15). The upshot of such work has been the design of more dislocation-resistant implants (data from this model were used to assist in the design of DePuy’s Pinnacle® Total Hip Section), the delineation of optimal orientations for surgical implantation of THA components, and the identification of relative risks of various patient activities believed to be responsible for dislocations. Recent FE work with THA dislocations (Figure 16) has involved embedding the implant components in anatomically realistic bone structures, and representing the hip capsule by means of hyperelastic elements whose input coefficients are set so as to mimic that tissue’s experimentally-observed stress/strain behavior (Stewart et al., 2004).
Case Study 2

The second case study also involves the hip, but addresses instead a naturally occurring disorder: femoral head osteonecrosis. Osteonecrosis is a skeletal pathology involving death of bone cells, usually with ischemia being the proximate cause. A wide range of etiologic factors is recognized, including trauma, alcoholism, corticosteroid regimens, blood coagulopathies, and various disorders of lipid metabolism. This condition is of interest biomechanically, because death of bone cells elicits a biological “repair” response that includes active weakening of the bone, due to enhanced activity of bone-resorbing cells (osteoclasts). Typically, as this structural weakening proceeds, the femoral head progressively collapses, over time periods of a few months to 1 or 2 years. Debilitating secondary osteoarthritis then
Figure 15 — Results from parametric exercise of the dislocation finite element model. In this series the ability of the total hip implant to resist dislocation was assessed for motion inputs characteristic of leg crossing in an erectly seated position, for a total hip acetabular component implanted at 20° of anteversion (i.e., rotation in the transverse plane), for various angles of abduction (tilt in the coronal plane) ranging from 30° to 70°. The ability to conduct such tests to isolate the effects of a single variable (here, acetabular component abduction) on an outcome measure of interest (here, peak resisting moment) is a major attraction of finite element analysis.

Figure 16 — Incorporation of capsule restraint and anatomic bony bed encasement in a 3-D contact finite element analysis of total hip dislocation. In this formulation, the capsule is characterized as a formal 3-D materially nonlinear region capable of arbitrary wrap-around contact with both the implant and the bony members. The locus of anatomic capsule attachment on the respective bony members is depicted by the thick solid lines.
develops, due to the resulting articular surface incongruity. The only recourse at that point is total hip replacement. Unfortunately, osteonecrosis patients tend to be young and active, and they have an extremely high incidence of prosthesis loosening.

Osteonecrosis in the U.S. accounts for about 10% of the total hip patient population, and for about 20% of that procedure’s aggregate societal cost. For this reason there is great interest in finding ways to forestall structural collapse of the natural femoral head. Finite element modeling has a role to play in that regard, since it provides a basis for assessing which aspects of disease involvement influence the speed and severity of collapse, and it allows quantifying the degree of improvement of the adverse biomechanical environment afforded by structurally-motivated surgical interventions.

Meshing in this instance requires an anatomic image source. Four modalities have played a role in providing source data for that purpose: serial physical sectioning, computerized tomography (CT) imaging, plane film radiographs, and magnetic resonance (MR) imaging. Physical sectioning affords the highest spatial resolution, at least in the specific sections photographed, but the sectioning procedure is cumbersome, and being destructive, it obviously is applicable only to cadaver specimens. CT scans hold the advantages of being available nondestructively, plus providing relatively good spatial resolution (voxel edge dimensions of typically 1 mm or so) for contemporary images of whole major articular joints.

Another attraction of CT images for FEA of bone is that, besides geometry, the voxel source data are in the form of Hounsfield numbers, an index of local radio-density, and therefore are convertible, usually by linear regression (Ciarelli et al., 1991), to physical density and hence elastic modulus. In the case of FE modeling of osteonecrosis, an idiosyncratic situation where the size, shape, and position of the necrotic lesion are key determinants of collapse propensity, obviously it is important to delineate the zone of involvement. MR scans are invaluable for that purpose, since one of the early aspects of the pathogenesis is saponification of the marrow of the involved cancellous bone region, for which the changes in water content lead to characteristic changes in MR signal intensity. MR is so sensitive for that purpose that it has become the clinical standard when osteonecrosis is suspected. This is both good and bad for purposes of FE modeling. Since the information provided by MR is so definitive for diagnosis, CT scans are generally not indicated clinically for these patients.

Conventional clinical MR sequences unfortunately do not perform well for depicting cortical bone, so the best information available for osseous geometry is usually just plane film radiographs. A hybrid approach has therefore evolved for constructing FE meshes for studying osteonecrosis: a generic density distribution template for a normal proximal femur based on CT data, scaled appropriately on the basis of patient-specific x-rays, and with the patient’s area of lesion involvement measured from MR images. Source data for material property attenuations in necrotic lesions have had to come from physical compression testing of samples (5-mm cubes) excised from pathology specimens (Brown et al., 1981).

Different mesh zoning strategies have proven appropriate for different purposes of FE modeling of osteonecrosis. One approach, for sensitivity studies exploring generic influences of lesion size/shape/location, has been to subdivide the proximal femur into serial transverse slabs, each such layer being further subdivided into concentric rings of (primarily hexahedral) elements (Figure 17a). Then, by considering specific subsets of these fixed elements to undergo material prop-
Figure 17 — Various types of mesh zonings useful for studying the biomechanics of osteonecrosis. (A) A 3-D mesh was built up from stacked layers of hexahedral elements, arranged with polar connectivity in the individual layers. This arrangement was based on corresponding serial physical sections cut from a cadaveric specimen, which were photographed and stylus-digitized. Regions of osteonecrosis are shown shaded. (B) The coordinates of selected nodes near the articular surface were manually edited in order to spatially associate specific subsets of elements with the anatomic subchondral plate and its environs. This meshing approach in turn allowed querying the model for the structural effects of specific features of subchondral plate and peri-subchondral bone pathology that are utilized clinically for disease staging. (C) Parametric variation of positioning of a drill tract (axis P1-P2) were used to study structural influences of core drilling and bone grafting, by manually editing nodal coordinates of an existing internal mesh so as to achieve element boundaries aligned with the surface of the drill tract. (D) An alternative approach, in which the connectivity for internal meshing is based on the graft itself, with parallel layers of concentric hexahedral element rings mapped to the periosteal surface of the bone.
erty attenuations representative of osteonecrosis, it is possible to systematically survey the effects of individual characteristic parameters. Another meshing approach, useful for investigating juxta-articular load transmission implications of subchondral plate involvement and (pathognomonic) subchondral fracture, has been to adopt an ad hoc tangential element layering structure, with very high spatial resolution periarticularly, and much lower resolution elsewhere (Figure 17b).

A third approach, used to investigate the mechanical implications of therapeutic core drilling and/or structural bone grafting, has been to manually edit the nodal coordinates of a preexisting serial-transverse-plane meshing structure, so as to align element boundaries with the material discontinuities introduced by the surgery (Figure 17c). This strategy allowed systematic study of core/graft positioning variants, in the interest of providing objective, mechanically-grounded surgical guidelines. Of course, manual editing of preexisting 3-d FE meshes is a labor-intensive endeavor, and also has limited generality for studying surgical interventions such as bone grafts or implants, since oblique intersections between mesh planes and surgically-induced material discontinuities can sometimes lead to highly elongated or highly oblique elements, which are known to perform poorly. (Most commercial FE programs have report flags that warn the user when aspect ratios of zoned elements exceed generally accepted thresholds. Often the preprocessing algorithms of these programs treat excessive violations of those good-practice norms as a formal error, explicitly terminating program execution.)

An alternative approach is intervention-based meshing, where characteristic geometric features of the induced material discontinuity, rather than of the undisturbed original anatomy, are used as the basis for structuring the mesh. An example is graft-based meshing to study fibular grafting (Figure 17d). Here the mesh consists of serial layers of concentric elements, but the element layering is perpendicular to the axis of the graft. This allows concentric element rings in each layer, with element boundaries aligned with the interface between the graft and the host bone. Each such graft-based element layer terminates peripherally at its bounding planes’ intersections with anatomic periosteal surface, the detection of which required special preprocessing interpolation logic. Major commercially available preprocessing programs (PATRAN in this instance) include capabilities for the user to write special command-language instruction sequences, to implement mathematical operations of various sorts as part of the process of automated and/or interactive mesh generation.

Another approach to meshing also merits mention: known as adaptive meshing, it involves solution-contingent alteration of the geometry (and/or, more generally, also of the material property distribution) during evolution of a multistage FE simulation. As applied to osteonecrosis, an example has been the use of structural optimization analysis, to find the particular orientation and positioning of a bone graft that provides the best possible load protection for a given lesion. This is implemented (Figure 18) by subsectioning the structure into a small number of super-elements having generic interconnection patterns (i.e., shared nodes on the super-element boundaries), but within which the coordinates of individual elements can be conditionally changed.

Starting from a plausible “trial” position of the graft, positioning is systematically perturbed (requiring corresponding nodal position changes, and stiffness matrix recomputation) to find which positioning change offers the best incremental benefit in terms of reducing lesion stresses. This provides a basis for ever more
Figure 18 — Adaptive meshing, here shown for an analysis undertaken to identify mechanically optimal positioning for structural bone grafting in osteonecrosis. The proximal femur was zoned into a small number of super-elements (heavy black lines), within which the nodal coordinates of actual individual elements were then computed based on their relative positions \((\xi, \eta)\) within the respective super-elements. After a trial stress solution was obtained for a given provisional graft position, the graft tip and root position were selectively perturbed, the coordinates delineating the super-elements were modified accordingly, the individual nodal coordinates were recomputed, and corresponding perturbations of the provisional stress field were computed. This provided a basis for serial adjustments of the coordinates of the graft tip and graft root, to iteratively identify graft positions that were ever more favorable from a structural standpoint. The global stiffness matrix had to be recomputed each time the finite element mesh was adapted to accommodate a modified graft position.
favorable repositionings of the graft, until the FE model finally converges on a graft configuration from which no further lesion stress improvement accrues. Special-purpose adaptive meshing formulations such as this normally lie outside the template of standard capabilities provided by commercially available finite element and preprocessor programs. The analyst instead needs to write ad hoc program code to read the finite element results for a given FE solution time step, alter the nodal coordinates and/or material properties according to whatever adaptation rule is being implemented, and initiate the subsequent FE time step. This off-line interruption/adjustment of an ongoing multitime step FE simulation can be done manually by starting and stopping the finite element solution at successive time steps, or it can be done automatically by means of script programming at the operating system level (Maxian et al., 1996).

A relatively new approach to FE zoning, coming into increased usage in the field of biomechanics, is known as voxel-based meshing. Here, as the name implies, the vertices of a voxellated image (such as an MR or CT image) are con-

Figure 19 — A voxel-based finite element mesh for the emu proximal femur, viewed from anteromedially, from superiorly, and in midcoronal cross-section. The source geometric data were from a high resolution CT scan, which also allowed input of (Hounsfield) density-dependent spatial variation of the elastic modulus. The rectilinear voxels of the CT image were converted to hexahedral elements, by means of invoking one-to-one correspondence between image voxel vertices and finite element nodes. In subsequent preprocessing, the mesh was smoothed so as to achieve continuous distributions of computed stresses and strains on the external anatomic surfaces. Loading was via prescribed distributions of traction (normal stress) on the articular surface of the femoral head, data for which were obtained experimentally.
verted directly into finite element nodes, thus generating a cubically connected array of block-shaped hexahedral elements, corresponding 1:1 to the voxels of the source image. This approach is attractive in that it lends itself to fully automated meshing of arbitrarily complex geometries. Although the stair-step jaggedness unavoidably present at the edges of voxellated structures was originally a drawback (Keyak & Skinner, 1992), increases in computer speed and memory have permitted continued improvements in FE mesh resolution, to the point that the stair-steps have become virtually unnoticeable at the edges of contemporary global models. Indeed, some of the very largest FE problem sizes yet undertaken in biomechanics (van Rietbergen et al., 2003) have been zoned in this manner.

To date, most work in this area has been for linear problems, where a consistent pattern of local interconnection between elements facilitates specialized logic for solving extremely large systems of simultaneous linear equations. An example is a voxel-based mesh (Figure 19) of the proximal femur of the emu, a bipedal animal model that is useful for studying the biomechanics of osteonecrosis owing to its mimicking the collapse process seen in human patients. Nonlinear voxel-

Figure 20 — A voxel-based articular joint contact problem. Illustrates geometries of the peri-articular regions of the distal tibia and the proximal talus, extracted from a CT scan, which are used to study contact mechanics of the ankle. The source image data are in the form of a 3-D array of regularly packed hexahedral volume regions (voxels), whose spatial arrangement of vertices forms the basis for defining an analogous arrangement of element nodes for finite element analysis. For purposes of studying intra-articular contact, these (originally rectilinear) hexahedral elements were remapped so as to align with the (curvilinear) articular surfaces, to preclude kinetic artifact from the stair-step voxellation of the parent CT dataset.
based formulations have recently begun to appear (Grosland & Brown, 2002) to address special situations arising in biomechanics, such as for example patient-specific articular joint contact problems (Figure 20).

**Conclusion**

Thirty years have now elapsed since the first applications of FEA in biomechanics. There has been ever-accelerating growth in sophistication of the formulations employed, and ever-improving realism of the resulting simulations. These improvements have been powered by truly prodigious improvements in the capabilities of computing platforms. In the ever-changing world of computer technology, perhaps the one anchor of stability has been that the cost of implementing a given arithmetic operation has historically dropped by about 50% every 6 months. The ensuing unrelenting treadmill of hardware and software obsolescence is widely regarded as a much-more-than-acceptable price to pay for the ever-widening opportunities thus opened. Finite element analysis therefore remains a thriving area of growth in the field of biomechanics, with no end in sight.

**References**


Acknowledgments

Financial assistance was provided by NIH grants AR46601, AR48939, AR49919, and by DePuy, Inc.

The work depicted in the two case studies was performed in collaboration with Drs. Kelly J. Baker, Douglas R. Pedersen, Richard A. Brand, M. James Rudert, Jiro Sakamoto, Karen (Reed) Troy, Nicole M. Grosland, Christopher F. Scifert, Anneliese D. Heiner, John J. Callaghan, and Mr. Mark E. Nadzadi and Mr. Kristofer J. Stewart. Ms. Julie M. Mock and Mr. Thomas E. Baer kindly assisted with manuscript preparation.