

# Smooth surface meshing for automated finite element model generation from 3D image data

Steven K. Boyd<sup>a,b,\*</sup>, Ralph Müller<sup>a</sup>

<sup>a</sup>*Institute for Biomedical Engineering, Swiss Federal Institute of Technology (ETH), University Zürich, Zürich, Switzerland*

<sup>b</sup>*Department of Mechanical and Manufacturing Engineering, University of Calgary, 2500 University Drive, N.W., Calgary, Alberta, Canada T2N 1N4*

Accepted 13 March 2005

---

## Abstract

Finite element (FE) modelling based on data from three-dimensional high-resolution computed tomography (CT) imaging systems provides a non-invasive method to assess structural mechanics. Automated mesh generation from these voxel based image data can be achieved by direct conversion to hexahedron elements, however these model representations have jagged edges. This paper proposes an automated method to generate smoothed FE meshes from voxel-based image data. Mesh fairing processes are utilized that allow constraints that control the smoothing process, and are computationally efficient. Surfaces of the mesh on the exterior, as well as interfaces between two tissues, can be smoothed by varying fairing parameters and constraint criteria. The method was tested on a variety of real and simulated three-dimensional data sets, resulting in both hexahedron and tetrahedron meshes. It was shown that the fairing process is linearly related to the number of smoothing iterations, and that peak stresses are reduced in FE simulations of the smoothed models. Although developed for micro-CT data sets, this fast and reliable mesh smoothing method could be applied to any three-dimensional image data where node and element connectivity have been defined. © 2005 Elsevier Ltd. All rights reserved.

**Keywords:** Finite element modeling; Automated meshing; Bone; Microarchitecture; Micro-computed tomography

---

## 1. Introduction

The assessment of the mechanical properties of bone is of fundamental importance in bone related research, and finite element (FE) modelling has become a central tool for that purpose. A classical approach has been to use three-dimensional (3D) computed tomography (CT) data to generate FE models by conversion of voxel data to FE elements, and this approach is typically used to estimate global properties (Keaveny and Hayes, 1993). More recently, the development of high resolution CT ( $\mu$ CT) has enabled generation of FE models providing fine detail (on the order of 10  $\mu$ m spatial resolution) of

bone architecture—a determinant of bone ‘quality’ (Müller, 2003)—so that stresses and strains at the trabecular level can be determined (Boyd et al., 2002; Judex et al., 2003; Van Rietbergen et al., 1999). The enormous advantage of  $\mu$ CT-based FE models is they allow non-destructive assessment of mechanical properties at resolutions that were not previously possible using clinical CT scanners. The  $\mu$ CT-based FE models are used, for example, to predict fracture at the trabecular level in osteoporotic bone (Niebur et al., 2000; Pistoia et al., 2002) and are critical for research into mechanisms of bone adaptation (mechanosensitivity) by evaluating tissue-level stresses and strains (Brown et al., 1990; Huiskes et al., 2000).

The complex 3D bone architecture measurements provided by  $\mu$ CT present a challenge for FE meshing. User-interactive FE meshing is impractical, especially because studies typically involve many specimens. Therefore, automatic mesh generation by direct

---

\*Corresponding author. Department of Mechanical and Manufacturing Engineering, University of Calgary, 2500 University Drive, N.W., Calgary, Alberta, Canada T2N 1N4. Tel.: +1 403 220 4173; fax: +1 403 282 8406.

E-mail address: [skboyd@ucalgary.ca](mailto:skboyd@ucalgary.ca) (S.K. Boyd).

conversion to hexahedrons or tetrahedrons is commonly used (Müller and Rügsegger, 1995) as it is an efficient and accurate solution providing the mesh resolution is adequate (Ladd and Kinney, 1998; Niebur et al., 1999; Ulrich et al., 1998). Hexahedron-based models, with mesh sizes reaching 10's of millions elements, can be solved using specialized FE solvers (Van Rietbergen et al., 1995, 1999) developed specifically for these types of meshes. Unfortunately, although bone surfaces and bone-cartilage surfaces are smooth in nature, the hexahedron meshes cannot represent that smoothness. While hexahedron FE models can provide measures of tissue-level stresses and strains, the solution is least accurate at the jagged surface (Camacho et al., 1997; Guldborg et al., 1998)—precisely the region of interest, for example, in studies investigating the mechanosensitivity of bone (Huiskes et al., 2000), or the influence of bone-cartilage interface stresses on joint disease development (Brown et al., 1984). While mesh smoothing can be performed using a Laplacian smoothing operator to provide a smooth mesh surface (Merz et al., 1996), the process results in mesh shrinkage which can have an important impact on FE predicted bone mechanics because of the sensitivity of bone strength to the amount of bone (i.e., bone volume ratio), and element distortions introducing singularities in the FE analysis.

The purpose of this work is to present an automated method for smoothing FE meshes from 3D image data at interfaces (i.e., bone surfaces or interfaces between different joint tissues such as bone and cartilage) that is efficient and retains fidelity to the original FE mesh (volume conserving) and controls element distortion. This method is applicable to meshes of theoretically any element type (i.e., hexahedron, tetrahedron) and provides control of the smoothing process through simple constraints. Although developed here for  $\mu$ CT application, the same method can be universally applied to image data collected from other imaging modalities such as magnetic resonance (MR) imaging.

## 2. Methods

The algorithm presented is based on a recently developed computationally efficient fairing technique developed for smoothing polygonal surfaces (Taubin, 2000, 1995), and is extended here for volumetric FE meshes.

### 2.1. Signal processing

Signal processing methods can be applied to meshes (graphs) containing connected vertices  $V$  (nodes), where the signal at each vertex  $v$  is defined as the neighbourhood of connected vertices. Thus, neighbourhood symmetry exists where a vertex  $v_i$  has a neighbour  $v_j$ ,

and  $v_j$  is conversely a neighbour of  $v_i$ . The neighbourhood of  $v_i$  is denoted  $v_i^*$ , and both symmetric or non-symmetric neighbourhoods can be defined (discussed in further detail later). Based on the neighbourhood definition, a Laplacian smoothing operator can be defined for each vertex with weights  $w_{ij}$  for each neighbour, thus resulting in a set of displacements:

$$\Delta v_i = \sum_{j \in v_i^*} w_{ij}(v_j - v_i).$$

These displacements form the signal to which low pass filtering via Laplacian smoothing can be applied. The Laplacian transfer function is  $f(k) = (1 - \lambda k)$ , where  $\lambda$  is a scaling factor  $0 < \lambda < 1$ , this can be applied  $N$  times to apply a desired amount of smoothing. Although Laplacian smoothing is a linear operation and thus efficient for large meshes, it results in over smoothing and hence undesirable mesh shrinkage. The solution to this problem is the key of Taubin's fairing approach (Taubin, 1995) which is to modify the transfer function to  $f(k) = (1 - \lambda k)(1 - \mu k)$  where the term  $(1 - \mu k)$  results in an unshrinking step of magnitude  $\mu$ . Through careful selection of  $\lambda$  and  $\mu$  (Taubin, 2000), an effective band-pass (bp) frequency for the transfer function is set by applying  $N$  smoothing iterations and results in a low-pass filter that does not cause polygonal mesh shrinkage.

### 2.2. Smoothing constraints

Imposing constraints on the mesh smoothing process provides control where, for example, it may be desired that vertices at the bone surface or bone-cartilage interface be smoothed differently than the rest of the mesh. Using non-symmetric neighbourhoods for vertices, so-called hierarchical constraints can be applied. Imposing constraints involves first identifying and labelling vertices, and then defining a non-symmetric neighbourhood.

All vertices (nodes) in the 3D mesh are labelled in a hierarchical order where, for example, surface vertices are labelled hierarchically higher than interior vertices. Four classifications of vertices were defined: (1) fixed vertices, (2) interior vertices, (3) interface vertices and (4) surface vertices. The lowest in the hierarchy are the *fixed vertices* which do not consider positions of any other vertices during the smoothing process and are therefore unaffected by smoothing (i.e., no neighbouring vertices are defined). *Interior vertices* have neighbours with equal or lower hierarchies (i.e., may consider a fixed vertices as a neighbour), as do *interface vertices* that are located at boundaries between different tissue types such as bone and cartilage (i.e., may include *interior* and *fixed* vertices as neighbours). Finally, *surface vertices* are at the top of the hierarchical order, and includes neighbours of all other vertex types (Fig. 1).

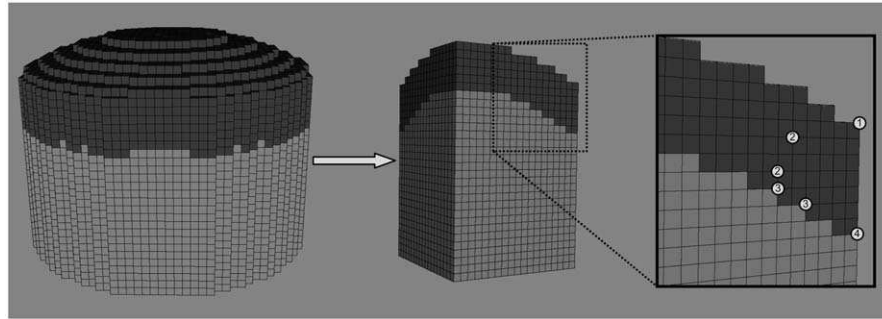


Fig. 1. Artificially generated data (left) representing a bone plug with a layer of articular cartilage is used to illustrate the selection of hierarchical constraints. Smoothing of the entire mesh is performed, including interior vertices as seen in the quarter section cutaway (middle-right). All vertices are classified (numbers 1–4) and subsequently appropriate neighbourhoods are defined as described in the text. The resultant smoothed mesh is shown in Fig. 7.

The hierarchically ordered vertices are used to define non-symmetric neighbourhoods that are the basis for controlling the smoothing process. For example, as a result of the non-symmetric neighbourhoods, the smoothing signal of *interior* vertices are dependent on all other vertices, but *surface* vertices (highest in the hierarchy) are dependent only on other *surface* vertices.

Classifying vertices in the FE mesh involves a straightforward topological analysis based on the valency (i.e., connectivity) of each node. Interior nodes have maximum valency (i.e., maximum valency in hexahedron meshes is six, and in tetrahedron meshes 20), surface nodes have less than maximum valency, and interface nodes are shared by elements representing different tissue types (i.e., bone and cartilage). Defining fixed vertices is at the discretion of the analyst; for example, it may be useful to limit any smoothing at a prescribed depth within the body of the FE mesh while permitting smoothing of surface elements.

The process of defining hierarchical constraints needs to be performed only once per mesh, and the mesh smoothing is performed for all vertices simultaneously with each smoothing iteration (a smoothed mesh is shown in Fig. 7). The number of defined non-symmetric neighbourhoods do not influence the computational speed of that process (Taubin, 2000).

### 2.3. Implementation and testing

The mesh smoothing methods described were implemented (C++ programming language), and the process of defining hierarchical constraints was fully automated as per the process described above. The program was tested on three data sets: (i) an artificial bone-cartilage plug to demonstrate interface smoothing between phases, (ii) a trabecular bone sample to demonstrate efficiency on large FE meshes ( $> 500,000$  elements), and (iii) an artificially generated sphere to demonstrate the reduction in peak stresses as a function of increased smoothing. Evaluation of the smooth

meshes include reports of mesh volume conservation, computation time, Gaussian surface curvature (a smoothness measure based on discrete data), and the Jacobian of elements to identify element distortions (i.e., a negative Jacobian indicates element distortion). Smooth meshes of the small data sets are solved using a commercial FE package (MSC Marc) with uniaxial compression boundary conditions applied (1% strain).

## 3. Results

FE meshes were generated from raw 3D  $\mu$ CT data automatically (Müller and Rüeeggger, 1995; Van Rietbergen et al., 1995) and with varying amounts of mesh smoothing on a modest PC Computer (Pentium III, 1.2 GHz, 640 MB RAM).

The computational time increases with the number of mesh smoothing iterations, and the relation between number of smoothing iterations and processing time is linear (Fig. 2a). The offset of this plot represents the overhead computational time ( $\sim 10$  ms) that includes defining the mesh based on 3D image data and defining hierarchical constraints. Similarly, computational time increased proportionally with the number of elements faired (Fig. 2b). Meshes of decreasing sizes were obtained by meshing smaller subvolumes of the trabecular bone rendered in Fig. 3 ( $1.7 \times 1.7 \times 1.7 \text{ mm}^3$ ). Meshes with over 1,000,000 elements (requiring approximately 265 MB RAM) were smoothed on a standard laptop. Hexahedron meshes took longer to smooth than tetrahedron meshes, but the difference was small (Fig. 2b).

A FE mesh generated from the trabecular bone sample was smoothed with varying numbers of iterations (Fig. 3). The smoothing was done with basic hierarchical constraints (differentiating only between surface nodes and interior nodes). A demonstration of applying further complex constraints where nodes lying at the extents of the trabecular bone image are fixed is

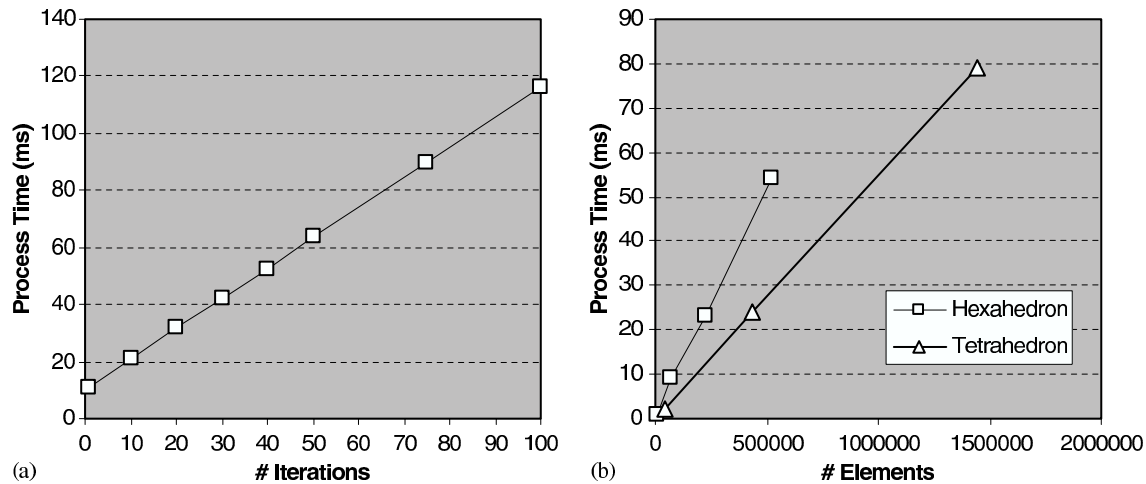


Fig. 2. The computer processing time (a) for smoothing a mesh with 525,000 elements as a function of the number of smooth iterations. The CPU clock time is linear with the number of steps, and the non-zero intercept is due to computational overhead. The processing time (b) increases proportionally with the number of elements in model while holding the number of iterations (five) constant. Hexahedron models take more computational time for smoothing than tetrahedron models.

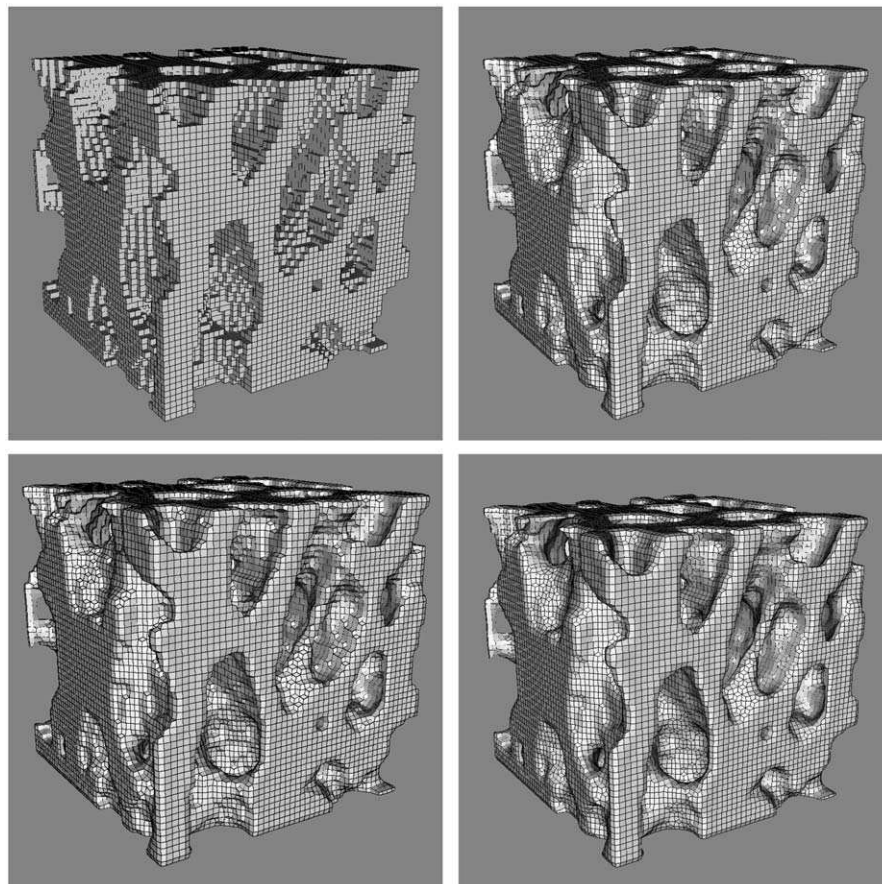


Fig. 3. A sample mesh representing a  $1.7 \times 1.7 \times 1.7 \text{ mm}^3$  sample of trabecular bone. The original mesh (top left) was smoothed with 2 (top right), 5 (bottom left) and 10 (bottom right) iterations. There is negligible visual difference for models smoothed greater than 10 iterations.

also shown (Fig. 4). The quantitative effect of the number of mesh smoothing iterations on element Jacobians and model surface curvatures (i.e., Gaussian

curvature as introduced previously) are presented for four different levels of mesh iterations (0, 2, 10, 20) (Fig. 5). The conservation of mesh volume for both

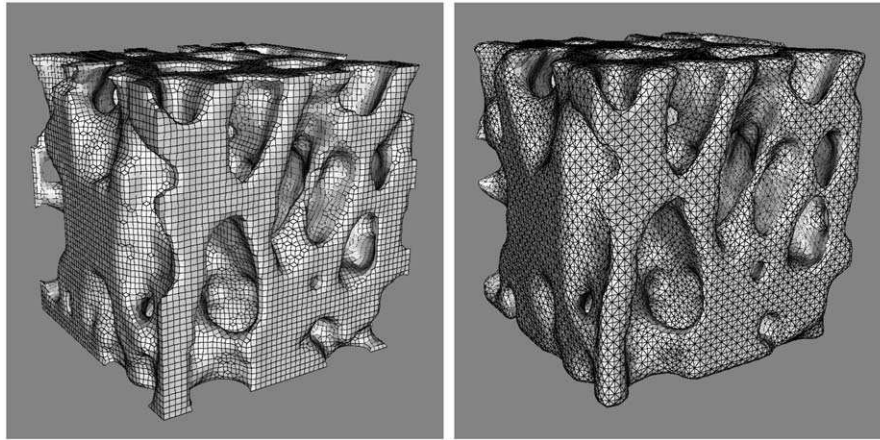


Fig. 4. A sample mesh representing  $1.7 \times 1.7 \times 1.7 \text{ mm}^3$  of trabecular bone after 20 iterations showing a hexahedron (left, 69,773 elements) and tetrahedron (right, 443,336 elements) mesh. The hexahedron mesh (left) also illustrates the application of constraints during smoothing where nodes falling on the image extent have been constrained (sharp edges), but in the tetrahedron mesh (right) no nodes are constrained (smooth edges). Constraint of edge nodes may be desirable to reflect a cleanly cut sample of trabecular bone for mechanical testing.

hexahedron and tetrahedron meshes (illustrated in Fig. 4) are presented as a function of the number of smoothing iterations (Fig. 6).

Illustrating the versatility of the mesh smoothing process, the artificial bone-cartilage plug was meshed with both hexahedron and tetrahedron elements. This data set also illustrated how data including two different tissue types can be smoothed at tissue boundaries (i.e., bone-cartilage interface). The use of interface vertices allows the bone-cartilage surface to be smoothed independently of neighbouring vertices thus providing a more realistic (smooth) boundary between the tissues (Fig. 7).

Using a commercial FE package (MSC Marc, Santa Ana, CA), representative meshes of both hexahedron and tetrahedron (Fig. 4) meshes were solved. Too much mesh smoothing can result in overly distorted elements (i.e., Jacobian close to zero, or negative) and it was found that the limit for smoothing iterations for the test models was eight for hexahedron meshes, and two for tetrahedron meshes. Illustrating the effect of the smoothing iterations on the FE stress distribution results, a sphere was meshed without smoothing, and with eight smoothing iterations. A sphere is a useful test of reduction of peak stresses due to the constant surface curvature on a perfectly (ideal) smoothed model. A decrease in the mean and peak element stresses resulted in the smoothed model (Fig. 8) compared to a traditional hexahedron voxel model with jagged edges.

#### 4. Discussion

Fully automatic generation of smooth surfaced meshes, with both hexahedron and tetrahedron elements, was achieved using the presented mesh fairing

algorithm. The degree of smoothing can be user controlled, and computational effort increases linearly with number of smoothing iterations. The  $\mu\text{CT}$  data, which is enormously complex in terms of both numbers voxels and representation of complex geometry, can be efficiently converted into smooth surfaced FE meshes with little or no deterioration in mesh volume.

Hierarchical constraints provide a powerful method to control the smoothing process without significantly adding to the computational complexity. In fact, more hierarchical constraints result in reduced mesh smoothing complexity because the number of node-neighbourhood searches is decreased. Interface smoothing (i.e., between bone-cartilage interfaces) is a particularly useful feature as it permits independent smoothing of that interface resulting in a more natural (i.e., smooth surface) FE mesh representation. Image data with segmented multiple tissues will occur with increasingly regularity as image modalities such as CT and MR are fully utilized to produce fused data sets (i.e., inter-modal image registration and segmentation). Another useful feature of hierarchical constraints is the possibility to control the depth of the elements that are affected by the smoothing iterations. Using a distance transformation algorithm, nodes can be labelled according to their distance from the mesh surface thus allowing the user to 'fix' nodes beyond a certain depth. Currently, large-scale FE solvers require that all elements have identical stiffness matrices (Boyd et al., 2002; Hollister and Riemer, 1993; Van Rietbergen et al., 1995), however if only elements near the mesh surface were smoothed and deeper interior elements were not altered, it may be possible to develop solvers that store a certain fixed number of stiffness matrices representing the smoothed surface elements, while applying a single stiffness matrix to the unaltered interior elements. This would may result

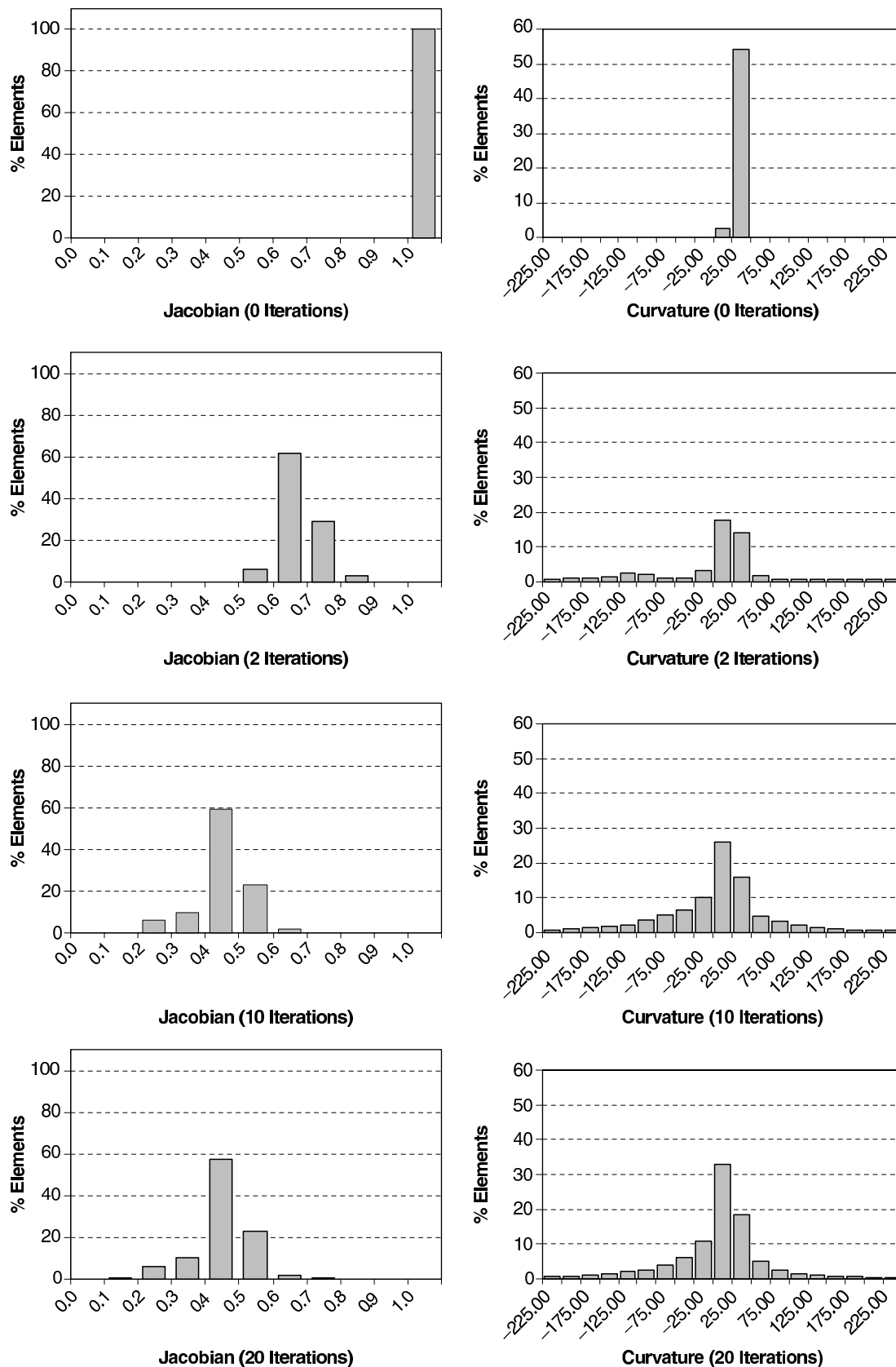


Fig. 5. The Jacobian (left column) and surface curvature (right column) is shown for a  $1.7 \times 1.7 \times 1.7 \text{ mm}^3$  sample of trabecular bone meshed with hexahedrons and smoothed with 0, 2, 10, and 20 iterations. After 20 iterations, the Jacobian remains above zero for all elements. On the range of curvatures included in the histogram, only 56% of the elements are captured in the  $\pm 225/\text{m}$  range with no smoothing (0 iterations; extremes in curvature occur in the jagged unsmoothed model). The percentage of elements (69,773 total) captured at 2, 10 and 20 iterations are 53%, 89% and 93%, respectively.

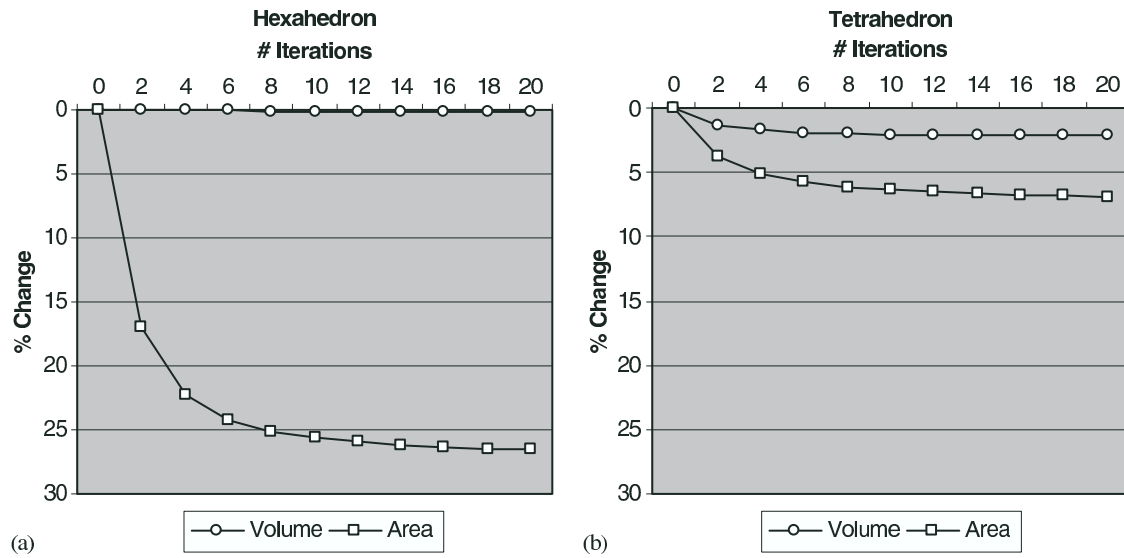


Fig. 6. Mesh volume ( $\text{mm}^3$ ) and surface area ( $\text{mm}^2$ ) for a  $1.7\text{ mm}^3$  sample of trabecular bone reported as a function of smoothing iterations for (a) hexahedron and (b) tetrahedron meshes. The maximum percent change in volume was a decrease of 0.09% and 2.2% for the hexahedron and tetrahedron meshes, respectively. Maximum percent change in surface area was a decrease of 26.6% and 6.9% for the hexahedron and tetrahedron meshes, respectively.

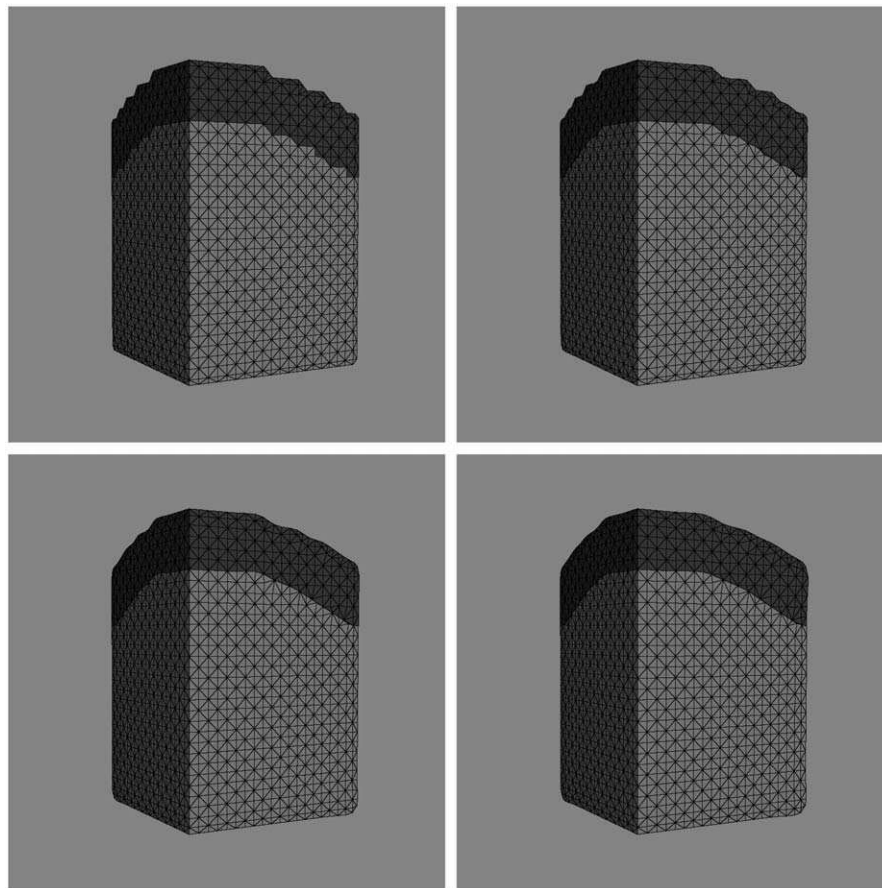


Fig. 7. A quarter section of a plug of bone (light gray) with a cartilage layer (dark gray) meshed with tetrahedrons (top left) and then smoothed with two iterations (top right), 10 iterations (bottom left) and 20 iterations (bottom right). Smoothing occurs on the outer surface of the plug (surface in profile) and at the bone–cartilage interface. All other nodes have lower hierarchical levels and therefore adapt to the smoothed positions of nodes with higher levels at the interfaces.

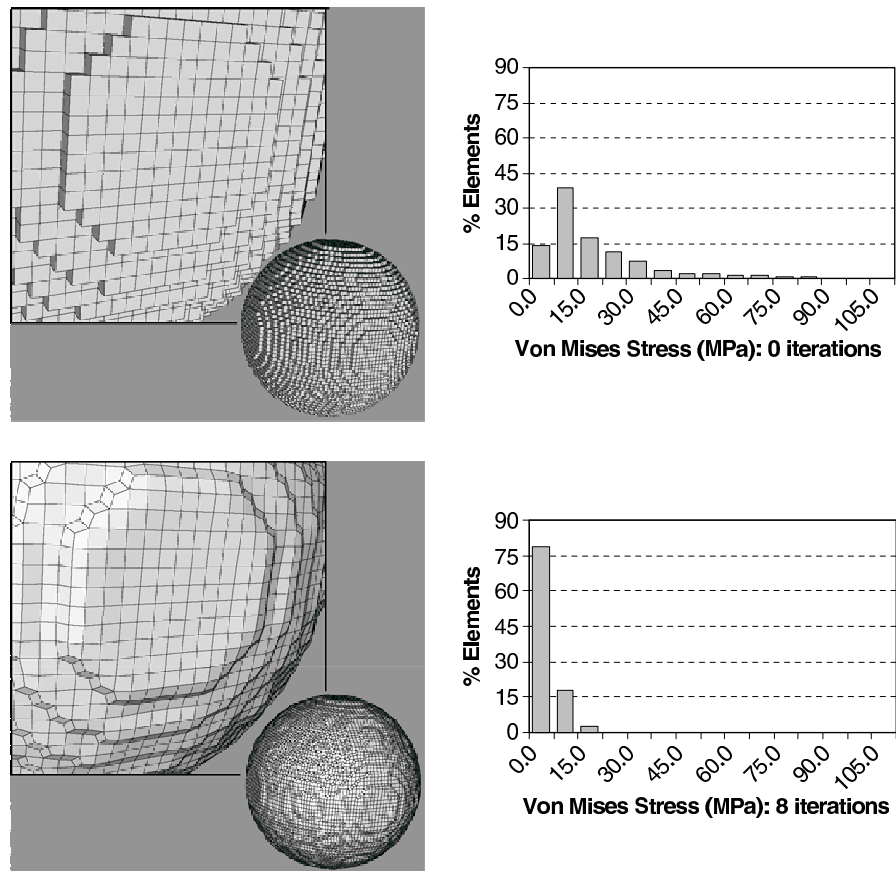


Fig. 8. A sphere was meshed without smoothing (top, with detail inset) and with eight smoothing iterations (bottom, with detail inset) and solved under conditions of uniaxial compression. The Von Mises stress is illustrated in the histograms based on the FE results. The mean stress was 20.1 and 4.9 MPa for the unsmoothed and smoothed models, respectively. The peak stresses decreased from 215.2 to 51.6 MPa, respectively.

in the necessary memory reduction to allow solution of large models, and provide a better (smooth) representation of cancellous bone. Efforts to develop such a solver are ongoing.

The mesh smoothing process has certain limitations that must be considered. It is evident that, although it can be applied to both hexahedron and tetrahedron meshes, it is better suited to hexahedron meshes. In terms of volume conservation (Fig. 6), the change in mesh volume for hexahedron elements was negligible which is an important feature of the mesh smoothing algorithm. The large decrease in mesh surface area that reaches an asymptote near 10 iterations is due to ‘flattening’ of the surface elements—jagged hexahedrons do not represent the bone surface well. In contrast to the hexahedrons, the tetrahedrons mesh volume decreased a small amount (i.e., approximately 2.2%), and stabilized after about six iterations. The decrease in tetrahedron surface area was less than for hexahedrons, and this is due to the intrinsically superior ability of tetrahedrons to represent a smooth surface (Müller and Rügsegger, 1995). The increased performance with hexahedrons compared to tetrahedrons was also evident in regards to the number of overly distorted elements, and the success

of solving these models using commercial FE software. An explanation for the problems with the tetrahedrons is that the valency (connectivity) of the nodes is high (i.e., up to 20) compared to hexahedrons (i.e., up to six), and increased valency is a known caveat of the fairing processing (Taubin, 2000, 1995). Future implementations of this mesh smoothing algorithm should incorporate constraints to prevent over-smoothing, and this could be implemented identifying distorted elements at each iteration (i.e., those having Jacobians below a pre-defined threshold) and fixing those element nodes to prevent further distortion.

The generation of automated FE meshes from  $\mu$ CT can be well controlled through a number of variables. First, as mentioned previously, hierarchical constraints can be applied in different combinations depending requirements of the individual data sets. Secondly, the mesh fairing can be controlled by setting the smoothing factor ( $\lambda$ ), unshrinking factor ( $\mu$ ), the effective pass-band frequency (pb), and of course the number of smoothing iterations. In practice, one should only control the pass-band frequency and smoothing factor  $\lambda$ , as the unshrinking factor  $\mu$  is dependent on those two values. These controls for the smoothing process have been

studied in depth (Taubin, 2000, 1995) and it is recommended to use a pass-band frequency in the range of  $0.01 < pb < 0.1$  and a smoothing factor of  $\lambda = 0.6307$  ( $pb = 0.1$  was found to be appropriate in this study). The modification of these parameters provide flexibility in the mesh smoothing process, and variables can be tailored for specific meshing requirements.

Accuracy of  $\mu$ CT-based FE meshes is partially influenced by mesh density and smoothness (Camacho et al., 1997). Unsmoothed hexahedron based FE models can result in stress raisers (Guldberg et al., 1998), and thus artificially high peak stresses, particularly at the bone surface. Smoothing the  $\mu$ CT-based hexahedron meshes reduces these artefact peak stresses, however rigorous tests for FE accuracy are still needed to be performed. Although it is not the ideal solution to generation of FE meshes from complex  $\mu$ CT data because it does not result in a reduction of elements, it is an efficient and fully automated method that requires no user interaction. The degree of smoothing is limited by distortion of the elements during the process, and future modifications of this algorithm should pay special attention to tetrahedron meshes to prevent distorted elements. A distinct possibility is to use thin-plate splines (Boyd et al., 1999) following a fairing process to ensure appropriate displacement of interior mesh nodes. Nevertheless, reasonable mesh smoothing was achieved for both hexahedron and tetrahedron FE meshes, in a fully automated and efficient algorithm, that can account for special circumstances such as inter-tissue interfaces. Although developed and presented here for  $\mu$ CT data, this method is applicable to data from any other imaging modalities that produce 3D image data.

## Acknowledgements

This study was funded by the Swiss National Science Foundation (FP 620-58097.99), Natural Sciences and Engineering Research Council (NSERC) Canada and Alberta Heritage Foundation for Medical Research (AHFMR). The authors would like to thank Renfeng Su for her technical support, and acknowledge the National Library of Medicine Visible Human Project for use of the head data.

## References

- Boyd, S.K., Ronsky, J.L., Lichti, D.D., Salkauskas, K., Chapman, M.A., 1999. Joint surface modeling with thin-plate splines. *Journal of Biomechanical Engineering* 121 (5), 525–532.
- Boyd, S.K., Müller, R., Zernicke, R.F., 2002. Mechanical and architectural bone adaptation in early stage experimental osteoarthritis. *Journal of Bone and Mineral Research* 17 (4), 687–694.
- Brown, T.D., Radin, E.L., Martin, R.B., Burr, D.B., 1984. Finite element studies of some juxtaarticular stress changes due to localized subchondral stiffening. *Journal of Biomechanics* 17 (1), 11–24.
- Brown, T.D., Pedersen, D.R., Gray, M.L., Brand, R.A., Rubin, C.T., 1990. Toward an identification of mechanical parameters initiating periosteal remodeling: a combined experimental and analytic approach. *Journal of Biomechanics* 23 (9), 893–905.
- Camacho, D.L.A., Hopper, R.H., Lin, G.M., Myers, B.S., 1997. An improved method for finite element mesh generation of geometrically complex structures with application to the skullbase. *Journal of Biomechanics* 30, 1067–1070.
- Guldberg, R.E., Hollister, S.J., Charras, G.T., 1998. The accuracy of digital image-based finite element models. *Journal of Biomechanical Engineering* 120 (2), 289–295.
- Hollister, S.J., Riemer, B.A., 1993. Digital image based finite element analysis for bone microstructure using conjugate gradient and Gaussian filter techniques. *Mathematical Methods in Medical Imaging II, SPIE* 2035, 95–106.
- Huiskes, R., Ruimerman, R., van Lenthe, G.H., Janssen, J.D., 2000. Effects of mechanical forces on maintenance and adaptation of form in trabecular bone. *Nature* 405 (6787), 704–706.
- Judex, S., Boyd, S.K., Qin, Y.X., Simmons, T., Ye, K., Müller, R., Rubin, C., 2003. Adaptations of trabecular bone to low magnitude vibrations result in more uniform stress and strain under load. *Annals of Biomedical Engineering* 31 (12–20).
- Keaveny, T.M., Hayes, W.C., 1993. A 20-year perspective on the mechanical properties of trabecular bone. *Journal of Biomechanical Engineering* 115 (4B), 534–542.
- Ladd, A.J., Kinney, J.H., 1998. Numerical errors and uncertainties in finite-element modeling of trabecular bone. *Journal of Biomechanics* 31 (10), 941–945.
- Merz, B., Niederer, P., Müller, R., Rügsegger, P., 1996. Automated finite element analysis of excised human femora based on precision-QCT. *Journal of Biomechanical Engineering* 118 (3), 387–390.
- Müller, R., 2003. Bone microarchitecture assessment: current and future trends. *Osteoporosis International (Suppl. 5)*, S89–S99.
- Müller, R., Rügsegger, P., 1995. Three-dimensional finite element modelling of non-invasively assessed trabecular bone structures. *Medical Engineering Physics* 17 (2), 126–133.
- Niebur, G.L., Yuen, J.C., Hsia, A.C., Keaveny, T.M., 1999. Convergence behavior of high-resolution finite element models of trabecular bone. *Journal of Biomechanical Engineering* 121 (6), 629–635.
- Niebur, G.L., Feldstein, M.J., Yuen, J.C., Chen, T.J., Keaveny, T.M., 2000. High-resolution finite element models with tissue strength asymmetry accurately predict failure of trabecular bone. *Journal of Biomechanics* 33 (12), 1575–1583.
- Pistoia, W., van Rietbergen, B., Lochmuller, E.M., Lill, C.A., Eckstein, F., Ruegsegger, P., 2002. Estimation of distal radius failure load with micro-finite element analysis models based on three-dimensional peripheral quantitative computed tomography images. *Bone* 30 (6), 842–848.
- Taubin, G., 1995. A signal processing approach to fair surface design. *Siggraph'95 Conference Proceedings*, pp. 351–358.
- Taubin, G., 2000. Geometric signal processing on polygonal meshes. *Eurographics: State of the Art Report*.
- Ulrich, D., van Rietbergen, B., Weinans, H., Rügsegger, P., 1998. Finite element analysis of trabecular bone structure: a comparison of image-based meshing techniques. *Journal of Biomechanics* 31 (12), 1187–1192.
- Van Rietbergen, B., Weinans, H., Huiskes, R., Odgaard, A., 1995. A new method to determine trabecular bone elastic properties and loading using micromechanical finite-element models. *Journal of Biomechanics* 28 (1), 69–81.
- Van Rietbergen, B., Müller, R., Ulrich, D., Rügsegger, P., Huiskes, R., 1999. Tissue stresses and strain in trabeculae of a canine proximal femur can be quantified from computer reconstructions. *Journal of Biomechanics* 32 (4), 443–451.