FINITE ELEMENT MESH GENERATION AND REMESHING FROM SEGMENTED MEDICAL IMAGES

Ashraf Mohamed  
Johns Hopkins University  
Department of Computer Science  
Baltimore, MD 21218, USA

Christos Davatzikos  
University of Pennsylvania  
Department of Radiology  
Philadelphia, PA 19104, USA

ABSTRACT

We present an approach for the automatic generation of patient-specific tetrahedral finite-element (FE) meshes from multiple-label segmented medical images. The approach uses a mesh refinement method with guaranteed tetrahedral element quality and includes a post-processing step with operations to change the mesh topology. Results indicate good approximation of the meshed geometry and acceptable simulation errors for a mechanical problem with a known analytical solution. We also present a method for correcting mesh distortion associated with large-deformation mechanical problems such as those that can arise in dealing with biological soft tissues. This adaptive remeshing scheme is driven by an estimate of the local \textit{a posteriori} FE error profile. Finally, we demonstrate the use of our approach to carry out a large-deformation simulation of non-infiltrating brain tumor growth starting from a segmented medical image.

1. INTRODUCTION

Patient-specific finite-element (FE) models constructed from three-dimensional (3D) medical images can be used to compensate for brain deformation during neurosurgery [1], predict deformation of the breast [2], estimate deformation of the heart’s left ventricle [3], or register a tumor-bearing brain scan to an anatomical atlas [4]. An important initial step in FE analysis is mesh generation, which involves the discretization of the multidimensional domain of the problem into small elements of simple geometry, such as tetrahedra or hexahedra for 3D problems.

To create patient-specific models, FE meshes should be generated automatically from the medical images. Most available automatic mesh generation tools target engineering design and analysis applications, and therefore are not, in general, well suited to deal with the triangular surface meshes typically used to represent objects extracted from medical images. Approaches that rely on constrained Delaunay triangulation (CDT) to generate tetrahedral FE meshes from triangular surface representations were proposed and integrated in some well known commercial FE programs [5]. However, in some cases, the constraints on the quality of the tetrahedral elements and the requirement that they conform to the input surface mesh may not be possible to satisfy simultaneously with CDT. User interaction may be required, and therefore, such approaches are not always automatic.

Many investigators attempted the creation of FE meshes from medical images. A group of such efforts, including our proposed approach, is characterized by an initial step of casting a regular grid of small cubes aligned with the axes of the images, and that cubes are eventually tesselated into tetrahedra [1, 6, 7]. Tetrahedra are popular for automatic meshing because they can be made to conform to the complex geometry of anatomical organs more easily than other 3D element types. Although suitable for some applications, the approaches in [1, 6, 7] may produce meshes with elements that are not good enough for accurate FE simulations, or that may not respect the geometry of problem.

We present an approach for automatically generating unstructured tetrahedral meshes from segmented 3D medical images satisfying requirements for accurate FE simulations. These requirements are explained in Section 1.1 and the details of our approach are given in Section 2. To vary the density of the mesh elements over the domain of the problem, an algorithm that produces a tetrahedral mesh with guaranteed quality for FE simulations is used. A post processing step including the use of topological mesh changes is applied to improve the quality of the resulting tetrahedral elements in the mesh. We also propose a remeshing solution to remedy the mesh distortion problem that arises in large-deformation mechanical FE simulations typical of biological soft tissues. Mesh distortion induces numerical instabilities which may force early termination of the simulation or cause convergence problems. The used adaptive remeshing scheme is driven by a \textit{a posteriori} estimate of the solution error to change the mesh density adaptively in order to improve the accuracy of the simulation.

In Section 3, we apply our approach to sample segmented medical images and show that the resulting meshes are good approximations of the geometry in the images and are suitable for accurate FE analysis. We report results for a validation experiment involving a mechanical simulation that employs a mesh generated by our approach. In addition, we demonstrate the ability of the remeshing approach to correct mesh distortion in order to complete a mechanical simulation of the growth of a large, non-infiltrating brain tumor in an MRI scan of a normal subject. The completion of this simulation was not possible without the use of remeshing. We conclude the paper with Section 4, in which we summarize the paper and present an outlook for future work.

1.1. Finite Element Mesh Generation Requirements

The domain of a meshing problem is a body \( B \subset \mathbb{R}^3 \). The main goal of a FE mesh generator is to create a mesh that conforms to the geometry of the meshed domain \([6, 8]\) (i.e., that is an accurate approximation of the geometry of \( B \)). A mesh must respect internal and external domain surfaces, therefore no element in the mesh can pierce or straddle those surfaces. In addition, topological compatibility conditions dictate that the mesh should have no overlapping or criss-crossing faces or edges, and that the domain must be uniquely and completely discretized [6].

Several factors affect the choice of the size of the elements to be used at a region of the domain. In general, high density of small elements is needed near surfaces of high local curvature (for accurate approximation of the geometry), or where the FE solution changes rapidly, while a lower density of larger elements is acceptable at other areas in order to reduce computational cost [8].

A \textit{sizing function} \( h : B \rightarrow \mathbb{R}^m \) may be defined over the meshed domain [9]. The size is a positive real value that, for example, denotes the maximum edge length of an element in the mesh. The values of \( h \) can be computed based on local surface curvature, \textit{a priori} knowledge of the FE solution, or by the user based on these
considersations [9]. The average local curvature of a surface, $r_c$, dictates a maximum length, $d_0$, for an edge that has both ends on this surface by

$$d_0 = 2r_c \sin(\phi/2), \quad (1)$$

where $\phi$ is a user specified maximum spanning angle, which is the angle at the center of curvature that subtends the edge lying on the surface.

Elements of a FE mesh should have neither overly large nor overly small internal angles [8, 10]. Such elements can induce large FE solution errors and cause slow convergence of the FE solver. Many measures for the quality of elements for FE analysis exist in the literature. Here, we adopt the quality measure for tetrahedral elements used in ABAQUS [11], our FE simulation environment:

$$q_t = V_t/V_s, \quad (2)$$

where $V_t$ is the volume of $t$, and $V_s$ is the volume of the equilateral tetrahedron that can be inscribed in the circumsphere of $t$. As a tetrahedron becomes degenerate, $q_t$ approaches 0.

2. METHODS

We present the details of our mesh generation approach then describe a solution to the mesh distortion problem that happens in large-deformation FE simulations. At the heart of our approach is the use of the two mesh topological operators, edge split and edge collapse [12]. Splitting an edge results in the subdivision of each tetrahedron incident on this edge into two smaller tetrahedra. Tetrahedra are subdivided to comply with the sizing function requirement or to make them conform to the geometry being meshed as explained below. Conversely, edge collapse is an operation that seeks the deletion of an edge from the mesh together with the ring of tetrahedra incident on that edge. This operation can help remove degenerate elements from the mesh.

2.1. Mesh Generation Procedure

Our approach for mesh generation is composed of five steps. The first two steps are similar to the approaches described in [1, 6, 7]. Initially, a regular array of small cubes is cast over the whole volume to be meshed. The size of these cubes is a user-specified parameter that determines the approximate size of the largest tetrahedron possible in the final mesh as will be clear below. Cubes entirely outside the meshed objects are discarded. In the second step, each of the small cubes is tesselated into five tetrahedra. Tessellation must alternate between two different configurations for adjacent cubes to satisfy the topological compatibility conditions [6].

In the third step, subdivision of the tetrahedra is carried out to satisfy the sizing function. Subdivision is performed using the longest-edge propagation path (LEPP) algorithm [13]. LEPP guarantees a strict lower bound on the quality of the generated tetrahedra [12]. In fact, using LEPP on the mesh of the previous step, only a finite set of tetrahedral shapes will result (seven, up to a similarity). The smallest quality factor associated with this set of shapes is $q_t \approx 0.23$. Such a bound on the quality of tetrahedra after refinement cannot be established by the methods in [1, 6, 7].

The fourth step deals with tetrahedra that are straddling internal or external surfaces of the meshed object after the previous step. These tetrahedra will be referred to as non-conforming tetrahedra. For a mesh to conform to the geometry of the meshed domain, all tetrahedra must lie completely on either side of domain surfaces. Therefore, every surface must be meshed by a triangular mesh whose facets are faces of tetrahedra in the volume mesh. Given this observation, any edge that is part of this triangular mesh must be associated with two tetrahedral faces that are on the surface (or more than two in case of intersecting surfaces). Thus, for a triangular facet that is already on the surface, we use an algorithm that seeks another triangular surface facet associated with each of the three edges of that triangle. In Figure 1, to make $BCDE$ conform to the geometry, node $D$ or node $E$ can be projected to the surface. Some of the tetrahedra in the mesh (not shown in the figure) may become degenerate or even inverted with either option. If this is the case, the algorithm attempts splitting edge $DE$ at $F$, its intersection point with the surface, or collapsing edge $DE$ to point $F$. Among those four options, the operation that produces the best-quality tetrahedra is chosen. The algorithm is started with a seed triangle that when moved to surface produces the least distortion among faces of all non-conforming tetrahedra. It continues recursively by investigating edges of triangles brought to the surface until all tetrahedra are conforming.

The final step is post-processing, which is necessary to improve the quality of tetrahedra that may have been compromised during the previous step. Smoothing or optimization of mesh node locations cannot help improve the quality of some meshes by itself [10], therefore a combination of edge collapse (to improve mesh connectivity), and node location optimization (to improve tetrahedral shapes) is used to reach a final mesh of acceptable quality. It is another advantage over post-processing approaches in [1, 6].

2.2. Remeshing and a Posteriori Error Estimation

In large-deformation Lagrangian FE simulations some of the elements of the mesh can become severely distorted. This negatively affects the accuracy of the simulation and may prevent its completion. Whenever this happens, it is necessary to remesh the problem, i.e., to replace distorted regions of the mesh with new ones that have good quality elements. In addition to overcoming mesh distortion, remeshing can provide a vehicle for utilizing a posteriori error estimates from the last simulation step to control the size of the elements in the newly generated mesh for improved accuracy in the next step. We adopt the approach of Molinari and Ortiz [12] which has the advantage of providing purely local a posteriori error estimates and therefore remeshing can be applied locally without need for changing the whole mesh. This error estimator can only be derived for a certain class of materials which includes linear elastic and some nonlinear elastic solids, and therefore is applicable to problems reported in this paper.

The contribution of a mesh element $e$ to the solution error is given by

$$I_e = \left( h^k \phi^h \right)^{1/k} \left[ \phi \right]_\Omega, \quad (3)$$

where $h^k$ is the size of element $e$, $\| \cdot \|_{k+1}$ is the Sobolev $H^{k+1}$ seminorm, $\phi$ is the computed deformation mapping by the previous FE simulation step, and $\phi^h$ is the restriction of $\phi$ to the inside of the finite element $e$. In order to reduce the purpose of remeshing, an element $e$ is subdivided if $I_e \geq T_e$, where $T_e$ is a refinement threshold. Subdivision is carried out using LEPP to ensure the quality of resulting elements. Using LEPP, the initial triangulation is subsumed within the refined triangulation, therefore this subdivision approach does not introduce projection or numerical diffusion errors in subsequent simulation steps. Similarly, an edge is collapsed if all tetrahedra incident on this edge satisfy $I_e \leq T_e$. With $T_e \ll T_i$, many of the elements of the mesh will remain unchanged. After subdivision and collapse based on the element error indicators, to ele-
3. RESULTS

The procedure mentioned above was used to create an automatic meshing and remeshing utility coded in C++ and dubbed TetSplit. In the input volume, a label is specified for each region that is homogeneous from the point of view of the physical model. Sizing function values are computed as the minimum of a user-defined size for each label and the estimated size using surface curvature as computed by (1), with the mean curvature of the local 3D surface estimated by a fitted quadratic patch [14] ($\phi = 20^\circ$).

3.1. Mesh Generation

To assess the accuracy of a mesh in approximating the geometry of a body, for every label $l$, we define the overlap percentage between the tetrahedral mesh and the segmented volume as

$$O_l = \frac{100 \times 2V_{mshl}}{V_{mesh} + V_{voxl}} \%,$$  \hspace{1cm} (4)

where $V_{meshl}$ is the volume of all tetrahedral elements classified with label $l$, $V_{voxl}$ is the volume of all voxels with label $l$, and $V_{mshl}$ is the overlap volume between the two.

Table 1. Mesh generation results for four segmented volumes. Volume 1 is a hand segmented CT scan of a brain tumor patient with three labels: brain, tumor and ventricles. Volume 2 is a segmented normal brain MRI with a separate label for the ventricles. Volume 3 is a segmented prostate MRI with a separate label for the central zone. Volume 4 is a synthetic volume of two concentric spheres with the central zone. Volume 4 is a synthetic volume of two concentric spheres. Volume 1 is a hand segmented CT scan of a brain tumor patient with three labels: brain, tumor and ventricles. Volume 2 is a segmented normal brain MRI with a separate label for the ventricles. Volume 3 is a segmented prostate MRI with a separate label for the central zone. Volume 4 is a synthetic volume of two concentric spheres with the central zone. Volume 4 is a synthetic volume of two concentric spheres. Volume 1 is a hand segmented CT scan of a brain tumor patient with three labels: brain, tumor and ventricles. Volume 2 is a segmented normal brain MRI with a separate label for the ventricles. Volume 3 is a segmented prostate MRI with a separate label for the central zone. Volume 4 is a synthetic volume of two concentric spheres with the central zone. Volume 4 is a synthetic volume of two concentric spheres.

In Table 1, we report results for meshes generated for four sample segmented medical images. An acceptable element quality for ABAQUS is $q_t \leq 0.02$. All resulting meshes had acceptable quality associated with their elements Post-processing is necessary to improve the quality of the mesh. For example, in Volume 4, post-processing improves the minimum quality from 0.0008 to 0.1077. However, post-processing is the most computationally demanding step, requiring between 34% to 70% of the total mesh generation times reported. Post-processing time depends on many factors including the number of nodes in the mesh, the surface representation, and the number of mesh nodes on the surface [S]. In particular, [S] affects the time of post-processing due to the surface reprojection necessary after unconstrained optimization for these nodes. Reported processing times are for a 1.13GHZ machine running the Linux operating system. Overlap percentages show good approximation of the geometry for labels of large volume (e.g. brain) and are slightly lower for other labels (e.g., ventricles and tumor). This is possibly due to the relative size of voxels, used to measure overlap volume, compared to the volume of the label which causes a higher measurement error. Due to space limitations, the surface meshes from only two of these datasets are shown in Figure 2.

3.2. Validation Using a Mechanical Simulation

The two concentric spheres mesh described above was used to simulate the expansion of the inner sphere, while applying a no-displacement boundary condition to the surface of the outer sphere. The mesh elements of the inner sphere were removed and a constant pressure was applied inside. A linear material was assumed for the region in between the surfaces of the two spheres. Under these conditions, (using a derivation similar to the one in [15]) the displacement of a point at radius $r$ from the center of the two spheres will be in the radial direction and can be computed analytically.

The maximum and root-mean-square (rms) errors over the mesh nodes on the surface of the inner sphere were 4.75% and 2.4% of the displacement there, respectively. Owing to the simple geometry of this problem, a similar tetrahedral mesh could be generated automatically using the mesh generator within ABAQUS. The maximum and rms errors in this case were 1.63% and 0.64% respectively. The results from both meshes were of the same order of magnitude, but the better accuracy of the ABAQUS mesh may be attributed to its perfect regularity. Both meshes use tetrahedral elements with quadratic shape functions.

3.3. Simulation of Brain Tumor Growth

We apply TetSplit in conjunction with ABAQUS, to simulate the deformation of brain tissue caused by the growth of a large, non-infiltrating brain tumor (e.g., meningioma). Realistic simulations of this deformation are needed to establish an image registration between a patient’s tumor-bearing images and brain atlases, thereby making the information in the atlases available for the brain tumor patient [4]. We adopt the assumptions of Kyriacou et. al [4] regarding the tumor growth process and make similar modeling decisions. Two limitations to the work in [4] can be dealt with using TetSplit. First, the simulations in [4] were performed on 2D brain images due to the absence of an appropriate means for automatic 3D mesh generation from patient images. Second, mesh distortion due to large-deformation induces numerical caused the FE simulation to terminate prematurely before reaching the target tumor size.

Starting with Volume 2 described above, a spherical tumor of diameter $6mm$ was added to the cortex, and a tetrahedral mesh was generated for the volume. An almost-incompressible Neo-Hookean homogenous material was assumed for brain tissue, with material constants that correspond to an initial Young’s modulus of $E = 18KPa$ and Poisson’s ratio $\nu \approx 0.475$. For the type of tumors of concern in this work, growth tends to be uniform in
all directions. Therefore, mesh elements inside the tumor were removed and pressure was applied gradually to the internal surface of the tumor to induce expansion. The choice of the pressure value influenced by the required final tumor size and the material properties. A no-displacement boundary condition was imposed on the outer surface of the brain to simulate the constraining effects of the dura and the skull.

In Figure 3, the simulation results are presented using a 2D slice of the volume. Applying a pressure of 20 kPa gradually to the inside of the tumor, the simulation terminates before reaching equilibrium because of severe mesh distortion near the tumor area. With one-time use of adaptive remeshing at pressure 10 kPa, the simulation could terminate at the equilibrium pressure. The final volume of the tumor at termination of the simulation was 18.7 cm$^3$ compared to 3.5 cm$^3$ without remeshing. Linear tetrahedral elements were used for this simulation. The initial mesh had 48,569 elements, with minimum quality of 0.0232. Before remeshing, 717 tetrahedra had $q_t \leq 0.02$. After remeshing, the number of elements was reduced to 45,590 due to edge collapse, predominantly in the regions far from the tumor. Ten elements had $q_t \leq 0.02$ after remeshing, with a minimum quality $q_t \approx 0.01$. This, however, did not prevent the completion of the simulation.

**4. SUMMARY AND FUTURE WORK**

In this paper, an approach for the generation of patient-specific tetrahedral FE meshes from multi-label segmented tomograms was presented. Special consideration has been given to shortcomings of mesh generation approaches similar to ours in the literature. By the use of LEPP the size of the mesh elements is allowed to vary over the domain while guaranteeing their quality for FE simulations. A combination of topological operations, with a method for projecting nodes to the surfaces of the meshed domain, makes the mesh conform to the geometry of the problem. A post-processing step including the use of topological operations eliminates degenerate elements from the mesh. To deal mesh distortion in large-deformation problems, a practical approach for adaptive remeshing using a posteriori error estimators was implemented and integrated in our mesh generator.

Mesh generation results were reported for sample segmented tomograms. The produced overlap ratios indicate good approximation of the geometry, and the statistics on the quality of the tetrahedra indicate their suitability for use in FE analysis. We reported the results of a FE mechanical validation experiment using a mesh generated by our approach. We also demonstrated the use of our approach for adaptive remeshing in the problem of mechanically simulating brain tumor growth. The completion of this simulation was not possible without adaptive remeshing.

Our presented approach cannot guarantee that final mesh edges will lie along surface features such as brain sulci, and therefore such features may be lost. Although Sullivan et al. [6] propose a method to move mesh nodes to single point features, an automatic selection of such features is not easy, and the approach may not readily generalize for line features or surface intersections. An approach that uses CDT locally may be necessary in this case and is currently being investigated. We are also investigating the use of other topological operations, such as edge flips [10], to attempt the elimination of a few degenerate tetrahedra that were retained after our approach for adaptive remeshing.

**5. REFERENCES**


