ABSTRACT

The finite element method (FEM) has established itself as a powerful tool in biomechanics. However, developing a finite element three dimensional mesh for irregular geometry object is still a labor intensive task hence limits the usage of the three dimensional analysis for dental structures. This study presented an automatic procedure to generate the three dimensional finite element mesh of a maxillary second premolar. Tooth embedding, image processing, three dimensional automatic meshing and convergence validation were the major phases of this study. Firstly, a second premolar was sliced and scanned parallel to the occlusal surface. A self-developed image processing system was employed to detect the boundaries of different material within each section. An automatic mesh generation program were employed on these boundaries to created tetrahedral elements based on a moving nodes of uniform cube approach. Six mesh models of the second premolar with linear and nonlinear element types were analyzed. Strain energy and vonMises stresses were reviewed for convergence in the crown regions. Results of the analysis indicated that this automatic meshing procedure can provide a feasible way to generate accurate three dimensional finite element mesh for dental biomechanics study.

INTRODUCTION

The finite element method is a powerful tool and theoretically can easily handle the irregular geometry, inhomogeneity, anisotropic material properties and complex boundary conditions of biological structures, hence, become a popular analytical method in biomechanics. Unfortunately, a major drawback of this method is the numerous amount of manual labor required to generate a three dimensional mesh model. Two dimensional meshes have been used extensively in early dental researches[1-2]. They were easy to build and also depicted the inhomogeneity and anisotropic of the dental structure. However, two dimensional approaches cannot simulate the realistic of clinical applications and the three dimensional complexity of the dental structures and the nature of stresses within the structures. For most three dimensional dental mesh models in recent researches[3-4], they were created by manual digitizing the nodal points on the dental boundaries that obtained from medical images. Nodes on the corresponding boundaries between consecutive slices were connected to form the volume elements. Special efforts and time have to spent in the connecting procedure in order not to create ill-conditioned elements for some immense geometrical alteration between slices. This strategy is not only causing a lot of time and efforts in building the mesh but also an error prone process. Consequently, developing an automatic tool to construct the three dimensional mesh of dental structure with good geometry and material properties approximation is become an important task. This study presents an automatic procedure to generate a three dimensional finite element mesh of the maxillary second premolar. Six mesh models of the second premolar were created by this automatic approach and the convergence validation were performed.

METHODS

Tooth embedding, image processing, automatic mesh generation and convergence evaluation were the major phases of this study. The detail process of each process was described follow:

Tooth Embedding and Image Processing

A freshly extracted intact maxillary second premolar was embedded in epoxy resin at a specific
orientation, and the outer surfaces of the resin block were machined into a cube. The cube was sliced serially to expose tooth-resin sections parallel to the occlusal surface (Fig. 1). Sections were scanned to transfer into digital image data (Fig. 2). A self-developed image processing system [5] was employed to detect the boundaries of enamel and dentine or dentine and pulp on each section image. Figure 3 shown the detected boundaries of the dental structure.

Automatic Mesh Generation

The mesh generation process was basically a moving nodes of uniform cube approach [6]. The boundaries of each section image were superimposed by a uniform grid points. For grid points that were within half grid space of the boundaries, they were moved to the nearest position on the boundaries. The grid points outside the enamel boundaries were discarded. The remaining grid points were then regarded as nodal points of the mesh within each slice. Employing the original connectivity between grid points, nodes on each slice can forms a two dimensional mesh with three or four nodes’ elements (figure 4).

Three dimensional elements were established by connecting the two dimensional elements between slices. The connection was worked in a grid cube fashion, i.e., two corresponding grid cells on two consecutive slices were connected to form a grid cube. Tetrahedral elements were generated from the grid cubes. Because enamel and dentin or dentin and pulp can coexist within a grid cube, to produce the correct interface geometry for different materials, nodal types within a cube need to be classified before tetrahedral element were created. The types of the eight vertices
of a grid cube can be classified as outside material A, on material A, inside material A, on material B, and inside material B as shown in Figure 5. Before breaking the cube into tetrahedral elements, the vertices and faces of a grid cube were assigned with integer numbers, and the two diagonal faces within the grid cube were also defined with mid-face numbers. Figure 6 shown the numbering of vertices and faces. The rules of generating tetrahedral elements were based on the separation form of the six outer faces and the two mid-faces of a cube. In order to maintain the elements continuity between adjacent cube, the separation form of a typical outer face was determined according to its cube number (the numbering sequence was from first slice down and on each slice from the upper left one then from left to right and up to down). Figure 7 shown the separation forms for the odd and even numbered cubes. But if the two diagonal vertices of a face were both on material A or on material B, the face was separated by connecting these two vertices and the breaking line was generated. Figure 8 shown one of these types of separation form for the mid-face7 and face1. After the separation forms of all faces were defined, tetrahedral elements were generated from the cube based on the breaking line of each face. But in some special cases, the separation forms of the eight faces (two mid-faces and six outer faces) were incompatible, i.e., there were no tetrahedral elements can be generated to cover the whole cube. In these special cases, a node at the center of the grid cube was created, and twelve tetrahedral elements were generated according to the separation forms of the cube’s six outer faces. Figure 9 presented one of these special cases.

OUT, node outside material A
ONA, node on material A
ONB, node on material B
INA, node inside material A
INPB, node inside material B
Convergence Evaluation of The Model

To evaluate the mesh model produced by this meshing procedure. The ANSYS (Swanson Analysis Inc., Huston, Pa.) program was employed to perform the stress analysis and convergence study. Earlier studies demonstrated that the effect of periodontal tissues on the stresses in the tooth were negligible above the cervicoenamel area[7-8], hence, the exterior nodes of the lower region (between section 13 and section 25 of figure 1) in the mesh were fixed in all directions as the boundary conditions. A vertical static load of 100N was applied on the lingual cusp tip (Fig. 10). The 100N load was determined from the normal chewing force as a third of the maximum biting force[8]. The isotropic material properties of enamel, dentin and pulp assigned in this model were given in table I. Six mesh models with three different element sizes (0.879mm, 1.074mm and 1.27mm) and liner (solid72) and nonliner (solid92) tetrahedral elements were analyzed. Strain energy of upper portion (between section 0 and section 11 in figure 1) and vonMises stresses were reviewed for convergence within all six models.

<table>
<thead>
<tr>
<th>Material</th>
<th>Young's modulus(E) N/mm²</th>
<th>Poisson's ratio()</th>
<th>References</th>
</tr>
</thead>
<tbody>
<tr>
<td>Enamel</td>
<td>84100</td>
<td>0.2</td>
<td>[8]</td>
</tr>
<tr>
<td>Dentine</td>
<td>18600</td>
<td>0.31</td>
<td>[8]</td>
</tr>
<tr>
<td>Pulp</td>
<td>2</td>
<td>0.45</td>
<td>[9]</td>
</tr>
</tbody>
</table>

RESULTS AND DISCUSSION

Figure 10 presented a mesh result of the maxillary second premolar. The time to generate this three dimensional mesh after all boundaries were established is less than 10 minutes on a Pentium PC with 16Mbytes of memory. Multiple contours of different materials within a section can be modeled fairly well as shown in figure 11. Table II presented the number of nodes and elements of the three different element size meshes. Quantitatively, the strain energies computed for the six models were shown in figure 12, which again indicates that convergence was achieved. (The differences between all models were within 10%). The convergence of the strain energy did not show a monotonic pattern is because that the geometry were different within meshes of different element size. For the convergence study, qualitatively, the vonMises stresses in one section (4mm below the tip) of the six models were compared in figure 13. The stress had the similar distributions and the values also achieved convergence. Results of strain energy and vonMises stress had
indicated the feasibility of this mesh strategy. Although the model with liner, 1.27mm element size mesh achieved the state of convergence, but, the geometry in the crown area might be distorted for these large elements. In order to simulate the geometry more accurately, element size less than 1mm is suggested to be used for more realistic analysis.

<table>
<thead>
<tr>
<th>Element size</th>
<th>Node number</th>
<th>Enamel</th>
<th>Dentin</th>
<th>Pulp</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.879mm</td>
<td>1419</td>
<td>1928</td>
<td>3861</td>
<td>132</td>
<td>5921</td>
</tr>
<tr>
<td>1.074mm</td>
<td>1015</td>
<td>1436</td>
<td>2611</td>
<td>80</td>
<td>4127</td>
</tr>
<tr>
<td>1.27mm</td>
<td>767</td>
<td>1049</td>
<td>1826</td>
<td>75</td>
<td>2950</td>
</tr>
</tbody>
</table>

![Graph showing s92 and s72 values for different element sizes](image)
CONCLUSIONS

Because of the versatility of the finite element technique, it has established itself as a powerful tool for biomechanics researches. This study provides a feasible method to generate three dimensional mesh for dental structures. With the tetrahedral elements, dental geometry and interface between different materials could be modeled more accurately with few ill-condition (large-distortion) elements. By employing this meshing tool, dental biomechanics should be simulated by FEA more easily. Different parameters of dental biomechanical problems could be investigated and better treatments for patient could be achieved.

REFERENCES