Individual Finite Element Model Based on the X-ray CT Data (Automated meshing algorithm adjusting to bony shape)

Norio INOU[1], Masaya JONISHI[1], Michihiko KOSEKI[1] and Koutarou MAKI[2]

[1] Department of Mechanical and Control Engineering, Tokyo Institute of Technology, Meguro-ku, 152-8552, JAPAN
[2] Department of Orthodontics, Showa University, Ota-ku, 145-0062, JAPAN

INTRODUCTION

Individual modeling method is a significant tool for examination of mechanical characteristics of a bone. The computational results provide useful information for diagnoses and medical treatments of illnesses of musculoskeletal systems. For this purpose, several methods to generate finite element models have been proposed.

The authors also reported on an individual modeling method based on the X-ray CT data [1]. The method automatically generates a finite element model with almost same size of finite elements. It is desirable to use small size of elements to express parts of a bone with small radius of curvature or small thickness. However, use of small size of elements leads to a huge model, which requires a lot of computational time on the simulation. This paper presents a meshing algorithm which adaptively controls element size according to shape of a bone.

MESHING ALGORITHM

As regarding expression of an object with finite elements, interpolation methods as typified by NURBS have a function to control size of the elements. These methods require geometrical information, however it is difficult to extract the information from the CT data.

We propose a new modeling method to control size of finite elements directly referring to CT data. The basic idea of the method is to express a bony shape with tetrahedral elements. The method is composed of four processes as shown in Fig. 1.

1) Extracting a voxel space of a bone from multi-sliced CT images.
2) Distributing nodal points in the space.
3) Generating elements by use of Delaunay triangulation.
4) Finishing the model by removing excessive elements.

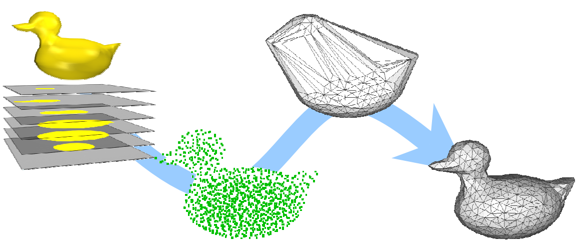

Fig. 1 Processes of individual modeling

To control element size according to shape of a bone, we introduce a "form factor" as the following steps.

Step 1: Counting number of subsistent voxels Nv around a remarking voxel in a cubic inspection space of which side length is n as shown in Fig. 2.

Step 2: Computing a form factor Vs as following equation.

$$Vs = \left| Nv - C \right| = \left| Nv - \frac{n^3}{2} \right|$$

Here, C is a criterion measure to compute the form factor, which is set to be half value of the cubic space.

Step 3: Distributing nodal points according to the form factor Vs. Nodal points are sparsely arranged at the portion where Vs is large. On the contrary, they are densely arranged at the portion where Vs is small.

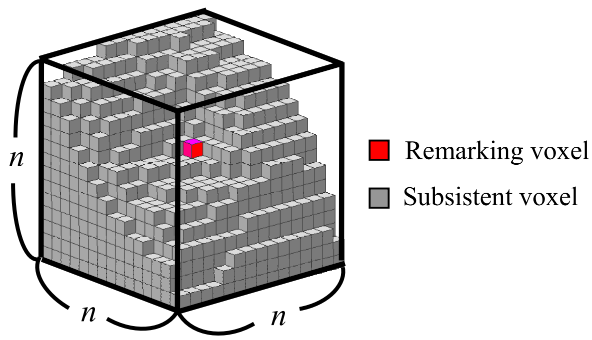

Fig. 2 Exemplification of form factor for size control

In the actual modeling process, size of each element is controlled in the three-dimensional voxel space. Here, we explain the arrangement of nodal points in the step 3 using two-dimensional space as shown in Fig. 3.

Figure 3 (a) illustrates a case that a part of the object has a sufficient thickness and a smooth surface. The inspection space around a remarking voxel in the center has an area of n x n lengths. In this case, the number of subsistent voxel Nv becomes 149. And the criterion measure C is $n^2/2 = 144.5$. Therefore, the form factor Vs is calculated at $|149-144.5| = 4.5$. In the same way, the form factors are calculated at every surface voxel of the object. Table 1 summarizes the number of subsistent voxel Nv and the form factor Vs in each case as shown in Fig. 3 (a)~(d).

Stress concentration may occur at the portions which have small radius of curvature (Fig. 3 (b) or (c)) or have thin thickness (Fig. 3 (d)). In such cases, the form factors have high values. On the contrary, stress concentration may not relatively occur at the portion with flat surface and sufficient thickness as in Fig. 3 (a), and the form factor has small value. Using the relationship between the form factor and shape of the object, we obtain the basic principle to control size of elements as described in the step 3. Thus the proposed method automatically controls size of finite element according to shape of the object.

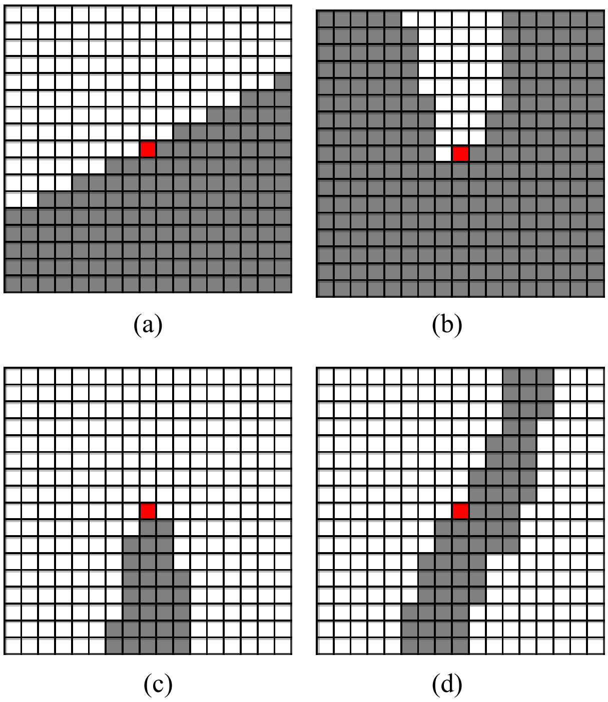

Fig. 3 Examples of partial shape of the object

Table 1 Subsistent voxels and form factor

	(a)	(b)	(c)	(d)
Subsistent voxels (*Nv*)	149	252	31	53
Form factor (*Vs*)	4.5	107.5	113.5	91.5

APPLYING TO A BONE

The proposed modeling method was applied to a human mandibular bone with the multi-sliced CT data. Figure 4 (a) shows the finite element model generated by our previous method. The model is composed of almost same size of finite elements. Figure 4 (b) shows the model with the proposed method. In this modeling, the ratio of the maximum and minimum element size was set to 10:1.

In the meshing process, if there is quite different in size between neighbor elements, bad-formed elements might be generated. To avoid generation of the bad-formed elements, we gave form factors to the inside object reflecting the form factor at the surface of the object. This process successfully generates a finite element model so that element size is smoothly changed all over the model. The model (b) has smaller elements at the high curvature portions such as condyles or coronoid processes compared with the model (a). Contrary, larger elements are arranged to the model (b) at flat and thick bony portions. The total number of nodes or elements of the two models are about the same as in Table 2.

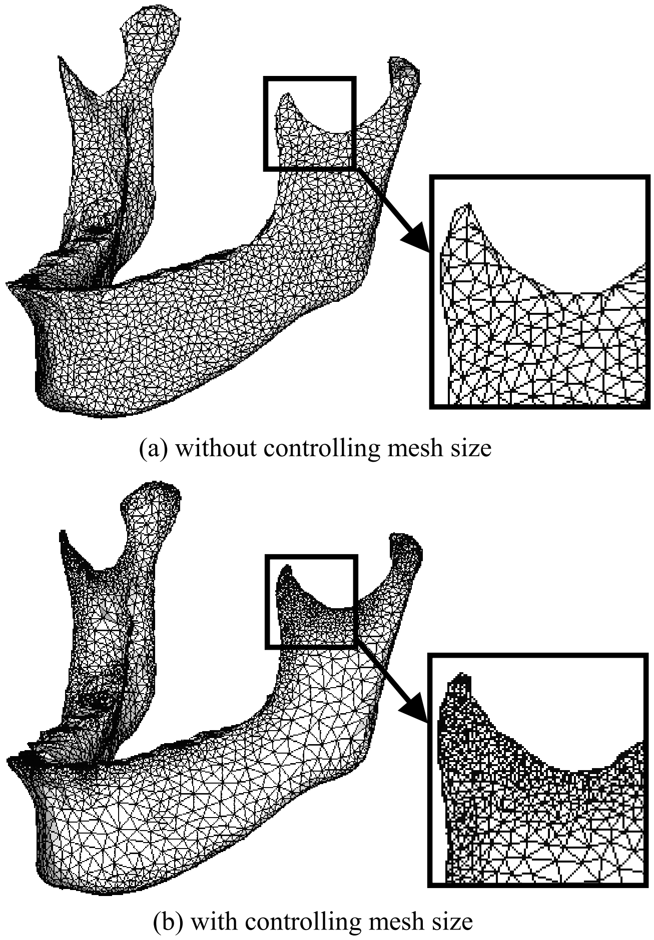

Fig. 4 Finite element models of human mandible

Table 2 Number of nodes and elements

	(a)	(b)
Number of nodes	10,646	9,962
Number of elements	53,083	44,529

CONCLUSION

A new meshing algorithm was presented for improvement of our individual modeling method. The algorithm controls size of finite elements using a form factor which is directly calculated from CT image data. The advanced individual method was applied to a human mandible. The generated finite element model well expressed the shape of the mandible under a limited number of nodes or elements.

The proposed method controls the mesh size based on distributed values directly calculated from the object data, so that it is also possible to control mesh size adjusting to bone density or stress distribution. A further direction of this study integrates the meshing algorithm considering these factors and demonstrates the effectiveness of the modeling method by stress analyses.

REFERENCES

[1] Inou et al., *J. Japan Soc. Mech. Eng. Series C*, 68, 669, 2002, 1481~1486. (in Japanese)

Corresponding author’s e-mail: inou@mech.titech.ac.jp